Return to Main Menu

PARAMETER LIST
ALARM LIST
M-CODE LIST

for

MAZATROL MATRIX

MANUAL No. : H740HA0030E

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. If such modifications are required, please contact the nearest Technical Center or Technology Center.**
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

01. 2006

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, solid check, 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 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 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 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}.

- 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 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 solid check 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.

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.

OPERATIONAL WARRANTY FOR THE NC UNIT

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 (0° 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.

CONTENTS

Page

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.

- NOTE -

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
D24	Number of revolutions during dwell at hole bottom for boring

Address (bit)	Outline
D25	Boring-bar tip relief
D26	Returning distance from hole bottom for boring or back-boring
D27	—
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
D34	—
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	—
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

Address (bit)		Outline
D52		Reduction ratio for the G00-based relief rate during a very-deep-hole drilling cycle
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		Circumferential 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 - D72		—
D73 - D77		Learning of cutting conditions (**DEP-Z** range)
D78 - D82		Learning of cutting conditions (**WID-R** range)
D83 - D90		—
D91	0	M04 is output/not output after the tool has dwelled at the hole bottom during a tapping cycle.
	1	The tool dwells/does not dwell after M04 has been output at the hole bottom during a tapping cycle.
	2	The tool dwells/does not dwell after it has been returned to the R-point during a tapping cycle.
	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.
	4	The finishing tool path is shortened/not shortened during a true-circle processing cycle (end milling).
	5	The tool path is shortened/not shortened during a true-circle processing cycle (chamfering).
	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.
	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.
D92	0	During a true-circle processing (end milling) cycle, E17 is used for axial feed or not.
	1	The R1-point height of the back spot facing is set to D1 or not.
	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.
	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.
	4	—
	5	—

Address (bit)	Outline
6	During planetary tapping, chips are ejected/not ejected automatically prior to the threading process.
7	—
D93	Unidirectional positioning for point-machining
D94	Unidirectional positioning for point-machining
D95	Auto-setting method for tapping
D96 - D144	—

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)
E3	—
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
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.

Address (bit)	Outline
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 during a helical approach scheme
E34	Element that automatically determines an approaching distance in a Z-direction tapered approach scheme
E35	Approaching gradient during the 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
E39 - E54	—
E55	3-D, Axial cutting-feed overriding
E56	3-D, Inversion check of curved-surface pattern
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

Address (bit)	Outline
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
E79 - E82	—
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)
E90	—
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
E98	Cutting method selection for end milling-mountain, pocket milling-valley unit
E99	Milling feed rate specification range for the shape sequence of the MAZATROL program

Address (bit)	Outline
E100 - E103	—
E104	Tool path selection
E105 - E144	—

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	Fixed value (0)
F5	Fixed value (0)
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	Circler cutting clamp speed coefficient for high-speed smoothing control
F10	—
F11	Vector constant for 3-D, tool-diameter compensation
F12	Return amount of pecking in drill high-speed deep-hole cycle or in G73
F13	Allowance amount of rapid-feed stop in deep-hole drilling cycle or in G83
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)

Address (bit)	Outline
F22	Deceleration area of automatic corner overriding (G62)
F23 - F26	—
F27	Handling of G92 (spindel speed clamp value) command at restart
F28	Threading chamfering angle
F29	Override value of automatic corner overriding (G62)
F30	G-code type selection
F31 - F39	—
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
F45	—
F46	—
F47 - F66	Common variable name
F67	—
F68	—
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

Address (bit)		Outline
F79	0	Holding of memory monitor address valid/invalid
	1	Selection of menu display
	2	Key history function valid/invalid
	3	Tool search method
	4	Selection of tap gear
	5	Display of tools currently in use valid/invalid
	6	Initial value of synchronous/asynchronous tapping during tapping tool registration
	7	Display of a MAZATROL monitor window valid/invalid
F80	0	MAZATROL function valid/invalid
	1	Automatic display of the navigation window on the occurrence of an alarm valid/invalid
	2	**MAINTENANCE CHECK** display at power on, displayed/not displayed
	3	Third page of the **MAINTENANCE CHECK** display, displayed/not displayed
	4	**GRAPHIC MAINTENANCE** display on the occurrence of an alarm, displayed/not displayed
	5	Learning of cutting conditions valid/invalid
	6	Editing on the **CUTTING CONDITION LEARN** display valid/invalid
	7	Destination of spare tool correction by the workpiece measurement
F81		Program management function
F82	0	Characteristics estimation result graph, displayed/not displayed
	1	Selection of inch/metric representation in **POSITION** display, **TOOL DATA** display, and **TOOL OFFSET** display modes valid/invalid
	2	Basis for tool life judgment
	3	—
	4	X-axis diameter display on the **POSITION** display valid/invalid
	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
	6	—
	7	—
F83	0	Output of the alarm history data as text data valid/invalid
	1	Operation record function valid/invalid

Address (bit)		Outline
	2	—
	3	—
	4	Program conversion type selection 1
	5	Program conversion type selection 2
	6	CMT/DNC input conversion type selection (only for lathe)
	7	—
F84	0	Fixed value (1)
	1	Fixed cycle (B → J)
	2	Spare tool search for EIA
	3	Timing to validate new workpiece offset data specified with a system variable
	4	Machine coordinate system (G92) selection
	5	Incremental/absolute data command in high-speed machining mode
	6	Tape operation, Not operated until the buffer is full/Operated at a unit of EOB
	7	When no tool data has been designated during EIA/ISO program execution with the MAZATROL tool length data validated, operation is executed/ alarm state
F85	0	Table rotational machining
	1	Radial interference check
	2	Type of coordinate system for controlling the tool tip point
	3	Tool tip point control scheme
	4	Fixed value (0)
	5	Reset to cancel G68.2, valid/invalid
	6	Display of surface definition θ
	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
	1	Milling-spindle start timing for a milling unit (with **MILL&TURN.** set under **TYPE** of UNo. 0)
	2	Override scheme for G0 during tool tip point control
	3	—
	4	Display of the **PART SHAPE** window

Address (bit)		Outline
	5	Override scheme for G1 during tool tip point control
	6	Selection of rotary axis reference position for tool tip point control
	7	Display format of **REMAIN** on the **POSITION** display
F87	0	Selection of whether or not the tool is to be offset by each change only in the deviation vector
	1	Selection of whether or not the check for mismatch of the workpiece origin and table rotation center is to be conducted
	2	Data alteration checking function valid/invalid
	3	—
	4	—
	5	—
	6	—
	7	—
F88		Set this parameter to specify functions related to the conversion from MAZATROL program into an EIA program.
F89		Set this parameter to specify functions related to the conversion from MAZATROL program into an EIA program.
F90		—
F91	0	In response to move command without decimal point, tool moves by 1/tool moves by 10
	1	Coordinate system shift using a MAZATROL program, valid/invalid
	2	Stroke inside check before movement/Stroke outside check before movement
	3	—
	4	Metric (Initial G20 is valid/invalid)/Inch
	5	In response to move command without decimal point:
	6	G00 interpolation/non-interpolation
	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)
	1	Modal at power-on or at reset (G17 or G19/G18)
	2	Fixed value (0), Dwell command always in time
	3	Tool-length compensation (G43 or G44) axis
	4	Tool-diameter compensation (G41 or G42) start up/cancel type

Address (bit)		Outline
	5	Tool-diameter compensation (G41 or G42) interference check
	6	Fixed-cycle hole-drilling axis
	7	Tool diameter compensation for an EIA/ISO program
F93	0	—
	1	Modal at power-on or at reset (G94/G95)
	2	Modal at power-on or at reset (G91/G90)
	3	Tool length of tool data for EIA/ISO program, valid/invalid
	4	Feed rate during machine lock
	5	Middle point during reference-point return
	6	Single-block operation mode at user macro operation instruction
	7	Fixed value (0)
F94	0	Movement to hole-drilling position in fixed-cycle mode
	1	External deceleration signal valid/invalid
	2	Tool length offsetting during G28/G30 execution, canceled/performed
	3	Modal at power-on or at reset (G01/G00)
	4	Tool command method using T-codes
	5	Fixed value (0)
	6	Fixed value (1)
	7	Tool offset amount effectuated in an EIA/ISO program
F95	0	Interrupt function using user macro instruction, valid/invalid
	1	Handling of macroprogram interruption and call
	2	Automatic return position to restart the program (Fixed to 1)
	3	G00 (positioning) command feed rate for dry run
	4	—
	5	—
	6	Manual-pulse interrupt amount cancellation with reset key, valid/invalid
	7	Coordinate system corresponding to G54 set with reset key, valid/invalid

Address (bit)		Outline
F96	0	Selection of variable number for tool offset amount
	1	Fairing function valid/invalid
	2	Processing for arc command blocks in high-speed machining mode, nonuniform feed/uniform feed
	3	—
	4	Selection of a corner judgment criterion in high-speed machining mode
	5	Selection of a cutting feed clamping speed in high-speed machining mode
	6	Rotational axis shape correction valid/invalid
	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
F104		Fine spline interpolation curve error (Block including no inflection point)
F105		—
F106		—
F107		Small block judgment length
F108		Corner deceleration angle increment value
F109		—
F110		—
F111	0	Selection of display type of tapping tool in solid mode
	1	Use/disuse of dry run during thread cutting
	2	Use/disuse of feed hold during thread cutting
	3	Direction of rotation of the C-axis during C-axial threading with G01.1
	4	EIA tool command suffix valid/invalid

Address (bit)		Outline
	5	Tool correction amount selection for EIA/ISO programs
	6	Execution mode selection for a fixed turning cycle
	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
	1	Data handling on the milling tool of a group that has expired in tool life
	2	Data handling on the turning tool of a group that has expired in tool life
	3	Tool life management of the FLASH tool
	4	Tool life management — Life time
	5	Tool life management — Maximum available wear offset data X
	6	Tool life management — Maximum available wear offset data Y
	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
	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)
	2	Tool shape check during tool measurement, valid/invalid
	3	Moving axes by using G49 (tool length cancel) in G43 (tool length offset) mode, valid/invalid
	4	Selecting a rethreading function
	5	Output timing of a tool life alarm
	6	Initial setting of G53.5
	7	The life of the tool is judged/not judged from its machining count
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

Address (bit)		Outline
F122		Clamping speed for the threading cycle — Z-axis
F123		—
F124		Permissible data alteration amount 1 for input error prevention function
F125		Permissible data alteration amount 2 for input error prevention function
F126 - F132		—
F133		Pitch of tapping tool for display in detail in solid mode
F134		Thread depth of tapping tool for display in detail in solid mode
F135		Tool-drawing accuracy in solid mode
F136		Amount of offset for dummy workpiece shape in solid mode
F137		Number of jaws displayed in solid mode for No. 1 turning spindle
F138		Number of jaws displayed in solid mode for No. 2 turning spindle
F139		Angle offset for the jaws displayed in solid mode for the No. 1 turning spindle
F140		Angle offset for the jaws displayed in solid mode for the No. 2 turning spindle
F141 - F144		—
F145		Rapid feed override when data alteration is detected
F146 - F153		—
F154		Parameter for system internal setting Setting prohibited
F155 - F160		—
F161	0	Shape/wear offset number separation, valid/invalid
	1	Shape offset handling
	2	Tool offset timing
	3	Tool offset vector handling if reset function is executed
	4	Shape offset handling if offset number 0 is entered
	5	Simplified wear offset, valid/invalid
	6	Succession of Z/C-offsets when a MAZATROL program is called from an EIA program
	7	Succession of Z/C-offsets 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

Address (bit)		Outline
	1	Type of passage of tool tip point through singular point
	2	Chamfer/corner R-command address selection
	3	Fixed hole-machining cycle return selection
	4	6 digits in T-command for turning
	5	Use of the M Pro scheme as the method of selecting the Length correction axis bit
	6	MAZATROL program check for missing Z-offset, valid/invalid
	7	Encoder polarity selection
F163	0	Bar feeder scheduling function, valid/invalid
	1	Incorporation of wear offset data into the current position display in EIA/ISO program mode, valid/invalid
	2	Incorporation of wear offset data into the current position display in MAZATROL program mode, valid/invalid
	3	Position of thread turning tool nose on solid drawing
	4	Barrier check on solid drawing, valid/invalid
	5	Menu on the **DATA I/O** display (floppy disk), displayed/hidden
	6	Menu on the **DATA I/O** display (tape), displayed/hidden
	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
	1	X-axis movement to minus side during polar coordinate interpolation, enabled/disabled
	2	—
	3	C-axis indexing when EIA subprogram is called from MAZATROL program
	4	Modal or non-modal state of Q command in deep hole drilling cycle
	5	Conversion of tool set data for milling tool based on head swivel angle when G53.5 is commanded
	6	Behavior of automatic operation of an EIA program when Z-offset is not set
	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
	1	Type of wear offset indicated in the milling tool list on the **TOOL DATA** display
	2	ID No./Tool name selection on the **TOOL DATA** display
	3	—
	4	—

Address (bit)	Outline
5	—
6	—
7	—
F167	—
F168	—

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
I4	—
I5	Function for making the G0 speed variable, Variable override: Minimum value
I6	—
I7	Function for making the G0 speed variable, Variable control area
I8	—
I9	Function for making the G0 speed variable, Variable control area lower limit
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)
1	Manual zero-point return operation
2	—
3	—
4	—
5	—
6	Removal of control axes, valid/invalid
7	—

Address (bit)		Outline
I14	0	Mirror image with respect to the machine zero point, valid/invalid
	1	—
	2	User software limits (I2, I3) valid/invalid
	3	Tool-tip relief after spindle orientation during execution of G75, G76, G86 or point-machining (boring or back-boring), valid/invalid
	4	Direction of tool-tip relief after spindle orientation during execution of G75, G76, G86 or point-machining (boring or back-boring)
	5	—
	6	—
	7	—
I15 - I24		—

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 position specification code
SU11	Movement of axes during approach
SU12	Rotating position specified in the index 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 W-axis

Address (bit)	Outline
SU16	Movement to C-axis index swivel position when Z-offset scheme is used
SU17 - SU48	—
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
SU54 - SU96	—
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 groove cutting unit and grooving
SU105	Cut depth (diametral value) for final cut in thread cutting unit Cut depth (diametral value) for final cut in composite-type thread cutting cycle G276, G76
SU106	Minimum cut depth clamping value in thread cutting unit and composite-type thread cutting cycle
SU107 - SU152	—
SU153 0	M-code selection for tapping cycle
1	—
2	—
3	—
4	—
5	—
6	—
7	—
SU154 - SU168	—

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 escape pattern from wall (90°) in rough cutting cycle
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)

Address (bit)	Outline
TC27	Recessing width for #1 to #3
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
TC35	—
TC36	—
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 edge clearance after roughing in the edge-machining unit
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 (**GRV**)
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

Address (bit)	Outline
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
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
TC60	—
TC61	Simultaneous operation pattern for transfer
TC62	Selection of tool change position specification code for FLASH tool
TC63	Amount of relief after transfer using the **TRANSFER** unit (Spindle mode 0 to 5)
TC64	Amount of relief after transfer using the **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 or in corner machining unit
TC68	Return distance in Z-axis at wall during rough cutting in bar cutting unit or in corner machining unit
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)
TC73	Return speed at pecking portion in groove cutting unit and turning-drilling unit
TC74	Pecking return distance in groove cutting unit and grooving (G274/G275)
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 (diametral value) for final cut in thread cutting unit Cut depth (diametral value) for final cut in composite-type thread cutting cycle G276
TC79	Minimum cut depth clamping value in thread cutting unit and composite-type thread cutting cycle G276
TC80	Angle of the tool nose during the G276 mode

Address (bit)	Outline
TC81	Final finishing repeat times in the composite-type fixed cycle (G276)
TC82	Chamfering data calculation coefficient in thread cutting unit and thread cutting cycle (G276/G292)
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
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
TC100	—
TC101	Selection of droop sampling axis
TC102	Selection of cycle counter sampling axis
TC103	Amplitude limit of table vibration
TC104 - TC110	—
TC111 - TC113	**CUTTING CONDITON LEARN** display — Workpiece length range
TC114 - TC116	**CUTTING CONDITON 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
TC119	—
TC120 - TC137	Distance to the front end of the long boring bar
TC138 - TC140	—
TC141 0	Use/disuse of acceleration in up-going slope during rough cutting cycle in bar machining unit
1	Use/disuse of deceleration in down-going slope during rough cutting cycle in bar machining unit
2	Selection between use/disuse of acceleration distance check at start of thread cutting unit
3	Selection between start position shift/start angle shift for thread number offset in thread cutting unit
4	Selecting an angle margin for nose shape compensation

Address (bit)		Outline
	5	Selecting an angle margin for nose shape compensation
	6	**CHUCK JAW DATA** display name/code selection
	7	Whether to make the partition plate and the workpiece barrier valid
TC142	0	Fixed value (0)
	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
	2	Selection of the jaw data reference method
	3	Using angle tool holder valid/invalid
	4	Selection of the method of moving axes to the tool change position
	5	Selection whether an alarm is to be issued if the ending position of workpiece pressing is reached during transfer of the workpiece
	6	X-axis retraction position during workpiece transfer as specified in the workpiece transfer unit of the MAZATROL program
	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
	1	Movement of the workpiece transfer axis for opposed turret machine
	2	C-axis clamping during workpiece transfer with C-axis positioning, valid/invalid
	3	Automatic output of spindle rotation command when turning tool is used in the MANL PRO unit
	4	—
	5	—
	6	—
	7	—
TC145 - TC154		—

7. SOLID (SD)

Address (bit)	Outline
SD1 - SD48	—
SD49	Machine coordinate selection
SD50	Table type
SD51 - SD96	—
SD97	Distance of model movement per time
SD98	Amount of model rotation per time
SD99 - SD124	—

2-2-2 Machine parameter

1. CALL MACRO (J)

Address (bit)	Outline
J1 - J40	G-code macroprogram call
J41 - J80	M-code macroprogram call
J81 - J90	—
J91 - J107	Parameter for system internal setting Setting prohibited
J108 - J144	—

2. MEASURE (K)

Address (bit)	Outline
K1	Rotational radius of the C-axis
K2	Minimum rotational angle
K3	Shaping control axis
K4 - K6	—
K7	Unbalanced axis
K8	—
K9	—
K10	Fixed value (0)
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)
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

Address (bit)	Outline
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	Predetermined value
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
K46	Excessive pressing error spread (Amount of drooping)
K47	—
K48	—
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

Address (bit)	Outline
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 ap-pointment call
K58	Type of T-code macroprogram ap-pointement 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	—
K69	G31.1 skip conditions
K70	G31.2 skip conditions
K71	G31.3 skip conditions
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)
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 - K84	—

Address (bit)		Outline
K85		Special linear acceleration/deceleration time constant for threading
K86 - K89		—
K90		Return override during synchronous tapping
K91		Alternative M-code for M96
K92		Alternative M-code for M97
K93		Fixed value (2)
K94		—
K95	0	—
	1	Fixed value (0)
	2	Tool position compensation during T-command execution, performed/not performed
	3	Coordinate system update during handle pulse interrupt, performed/not performed
	4	Fixed value (0)
	5	Acceleration/deceleration time constant for handle pulse feed
	6	Software limits for G30 execution valid/invalid
	7	In-position check valid/invalid
K96	0	G0 command in-position check valid/invalid
	1	Timing of manual free feed finish signal
	2	Fixed value (0)
	3	Axis/Cutting interlock alarm display, valid/invalid
	4	Suppression of lost motion in modes other than the G1 command mode, valid/invalid
	5	Fixed value (0)
	6	Fixed value (0)
	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

Address (bit)		Outline
K101		Dynamic offset, name of orthogonal axis
K102	0	—
	1	—
	2	Length of linear acceleration/deceleration filter II
	3	Length of linear acceleration/deceleration filter II
	4	Length of linear acceleration/deceleration filter II
	5	Length of linear acceleration/deceleration filter II
	6	Handling of the acceleration/deceleration filters for shape correction
	7	Fixed value (0)
K103	0	—
	1	Synchronous control error auto-correction valid/invalid
	2	—
	3	DDB micron unit
	4	Thermal displacement compensation interval
	5	Output of S-code and T-code at restart
	6	—
	7	—
K104	0	Output of a laser measurement B-axis 0-degree command after ATC, valid/invalid
	1	Direction of the laser axis of the laser measuring instrument (L16/K104 bit 2)
	2	Direction of the laser axis of the laser measuring instrument (Parallel to the Y-axis/X-axis)
	3	Vocal output function valid/invalid
	4	Type of voice
	5	Vocal guidance for warm-up operation
	6	Setting rapid feed override to 0% when cutting feed override is set to 0%
	7	—
K105	0	—
	1	Fixed value (1)
	2	S-code macro call invalid/valid

Address (bit)		Outline
	3	T-code macro call invalid/valid
	4	Second auxiliary function macro call invalid/valid
	5	Fixed value (0)
	6	Fixed value (0)
	7	Input in millimeter/inch
K106	0	Execution conditions for user macroprogram interrupt
	1	Start timing for user macroprogram interrupt
	2	Fixed value (0)
	3	Pitch error setting
	4	Fixed value (0)
	5	Fixed value (0)
	6	Fixed value (0)
	7	Fixed value (0)
K107	0	Fixed value (0)
	1	—
	2	—
	3	—
	4	—
	5	—
	6	Deceleration for arc valid/invalid
	7	Fixed value (0)
K108		Permissible error range for synchronous control
K109		—
K110		Judgment angle near a singular point
K111		Clamping speed in safety supervisory mode 3
K112		—
K113		Machine type

Address (bit)	Outline
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
K118 - K120	—
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
K129 - K144	—

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
L8	Skipping stroke limit for MMS
L9	Selection of random ATC specifications
L10	Interval between magazine pockets
L11	Touch sensor’s interference direction

Address (bit)	Outline
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	—
L18	Presence/absence of tailstock
L19	—
L20	—
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 escape from the approach point after TOOL EYE measurement
L29	Machine efficiency
L30	Selection of machining navigation case introduction messages
L31 - L36	—
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
L72	—
L73	Time constant for shape coorection 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	—
L79	In-position width for changeover of the synchronized-tapping gain
L80	—
L81	Fixed value (0)
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)
L89	Correction value of alignment deviation Y (90-degree face)
L90	Correction value of alignment deviation X (180-degree face)
L91	Correction value of alignment deviation Y (180-degree face)
L92	Correction value of alignment deviation X (270-degree face)
L93	Correction value of alignment deviation Y (270-degree face)
L94 - L97	—
L98	Max. tool length for laser tool length measurement
L99	Cycle time for saving the operational status management data
L100 - L105	—

Address (bit)		Outline
L106	0	Measuring equipment selection
	1	Selection of a rotational reference coordinate system for WPC-th
	2	Selection of measuring equipment
	3	Selection of whether workpiece measurement results and tool measurement results are to be stored into tool data of the lower turret
	4	Selection of whether to enable or disable the fixed amount compensation function
	5	Selection of measurement execution timing
	6	Selection of tool measurement operation
	7	—
L107	0	Tool path drawing
	1	—
	2	Tailstock type
	3	Whether tail thrust is also to be displayed in pounds (lbs)
	4	LBB No. setting for the grooving I.D. tool, threading I.D. tool, or touch sensor, valid/invalid
	5	XYZ-axis operation for the first T-command after cycle start
	6	A-axis operation for the first T-command after cycle start
	7	—
L108		Fixed value (0)
L109	0	Software limit 4 - Interference axial direction (1st set)
	1	Software limit 4 - Interference axial direction (2nd set)
	2	Software limit 4 - Interference axial direction (3rd set)
	3	Software limit 4 - Interference axial direction (4th set)
	4	—
	5	—
	6	—
	7	—
L110	0	Opposed-spindle lathe specifications
	1	Vertically inverted spindle specifications

Address (bit)	Outline
2	Axis name of the secondary spindle
3	Whether to disable or enable the display of “section to be machined” in the milling tool sequence
4	Display of the **BUFFER**, **REMAIN** and **POSITION** information during coordinate conversion
5	—
6	—
7	Z-axis direction
L111	—
L112	—
L113	Interference check reference axis (1st set of axes)
L114	Interference check reference axis (2nd set of axes)
L115	Interference check reference axis (3rd set of axes)
L116	Interference check reference axis (4th set of axes)
L117	Interference check axis (1st set of axes)
L118	Interference check axis (2nd set of axes)
L119	Interference check axis (3rd set of axes)
L120	Interference check axis (4th set of axes)
L121	Interference clearance (1st set of axes)
L122	Interference clearance (2nd set of axes)
L123	Interference clearance (3rd set of axes)
L124	Interference clearance (4th set of axes)
L125	—
L126	Positioning direction of the head rotation - α-axis for oblique plane indexing
L127	—
L128	—
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

Address (bit)	Outline
L132	Acceleration/deceleration filter (2nd stage), G0 time constant
L133 - L138	—
L139	Minimum usable tool diameter of the measurable chamfering tool
L140 - L144	—

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
M17 0	—
1	Unit of output from MCP to servo amplifier
2	Direction of machine zero-point return
3	Error correction schema with servo on
4	Type of axis

Address (bit)		Outline
	5	Rotational direction of servo motor (for movement in (+) direction)
	6	—
	7	Alarm/No alarm with axis removed
M18	0	Type of C-axis
	1	—
	2	Machine zero-point position
	3	—
	4	Watchdog-less axis
	5	X-axis current position display in radius/diameter
	6	Automatic/manual simultaneous absolute-value updating
	7	Absolute-value detection
M19	0	—
	1	—
	2	—
	3	In-position checking method
	4	—
	5	Homing operation starting position check
	6	—
	7	Backlash scheme to be adopted for watchdog-type returning to home position
M20	0	Rotational direction of the rotation axis (Dynamic offset II)
	1	—
	2	—
	3	—
	4	—
	5	—
	6	—
	7	—

Address (bit)		Outline
M21	0	Rapid-feed acceleration/deceleration type, Linear acceleration/deceleration
	1	Rapid-feed acceleration/deceleration type, First-order lag
	2	Rapid-feed acceleration/deceleration type, Second-order lag
	3	Rapid-feed acceleration/deceleration type, Exponential acceleration/linear deceleration
	4	Cutting-feed acceleration/deceleration type, Linear acceleration/deceleration
	5	Cutting-feed acceleration/deceleration type. First-order lag
	6	Cutting-feed acceleration/deceleration type, Second-order lag
	7	Cutting-feed acceleration/deceleration type, Exponential acceleration/linear deceleration
M22	0	Deceleration time constant for rapid-feed exponential acceleration/linear deceleration
	1	Type of stroke-end stop
	2	Type of stroke-end stop
	3	—
	4	—
	5	—
	6	—
	7	—
M23		—
M24		—
M25		Upper limit of inclined Y-axis access inhibition area (Soft limit 3)
M26		Lower limit of inclined Y-axis access inhibition area (Soft limit 3)
M27		Optimum acceleration control: Target speed
M28		—
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

Address (bit)	Outline
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 - M48	—

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)
N6	—
N7	OT time
N8	Creeping speed during initial zero-point return
N9	Amount of grid ignorance during initial zero-point return
N10	Grid spacing
N11	—
N12	Rapid-feed backlash
N13	Cutting-feed backlash
N14	—
N15	Width to which the machine posture change correction is to be applied
N16	Machine posture change correction value
N17	Servo amplifier channel number
N18	Servo amplifier rotary switch number

Address (bit)		Outline
N19		Axis system number
N20		—
N21	0	Linear-type rotational axis
	1	Rotational axis shortcut Invalid/Valid
	2	Fixed value (0)
	3	Bi-directional pitch error correction
	4	Reference axis for superposition control
	5	Superposition axis for superposition control
	6	Relative polarity of control axis
	7	—
N22 - N24		—
N25		Time constant for deceleration rate calculation
N26		Accuracy coefficient for deceleration rate calculation
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)
N36 - N48		—

6. ANOTHER (S)

Address (bit)	Outline
S1	—
S2	—

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
S9	—
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	Amount of reference position correction (Only for bidirectional pitch error correction)
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
S20	—
S21	—
S22	Cutting feed clamping speed during tool tip point control
S23	Reference workpiece zero point
S24 - S48	—

7. SPINDLE (SA)

Address (bit)		Outline
SA1 - SA8		Maximum RPM of spindle 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 RPM of spindle during tapping cycle (range 1 to 8)
SA25 - SA32		Spindle speed during gear shifting (range 1 to 8)
SA33 - SA40		Acceleration/deceleration time constant for the synchronous tapping (range 1 to 8)
SA41		Spindle orientating speed
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
	1	Spindle speed range changing method, in relation to switching the torque factors for auto-pecking of the cutting load detection type 2
	2	Homing direction for synchronous tapping
	3	Homing direction for synchronous tapping
	4	Defines the specified direction as the Z-phase detection direction
	5	Spindle index gear correction
	6	—
	7	—
SA46	0	Direction of orientation
	1	Direction of orientation
	2	C-axis position control changeover type
	3	Synchronous tapping position control changeover type
	4	Z-phase detection direction
	5	C-axis homing direction
	6	C-axis homing direction
	7	Synchronous tapping command polarity
SA47	0	—

Address (bit)	Outline
1	—
2	—
3	—
4	—
5	—
6	—
7	Considering/Ignoring the spindle/motor gear ratio
SA48	Encoder signal input destination
SA49	Speed attainment detection width
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	Revolutions in the following spindle output diagrams: MACHINING NAVIGATION-RESULT/MACHINING NAVIGATION-PPEDICTION/Monitoring Functions

Address (bit)	Outline
SA74 - SA80	Output in the following spindle output diagrams: MACHINING NAVIGATION-RESULT/MACHINING NAVIGATION-PPEDICTION/Monitoring Functions
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
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
SA98	—
SA99	Orientation time constant
SA100 - SA113	—
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

Address (bit)	Outline
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
SA129 - SA137	—
SA138	Number of spindle gears
SA139	Number of motor gears
SA140	Turret indexing gear ratio
SA141 - SA143	—
SA144 0	—
1	—
2	—
3	—
4	—
5	—
6	Spindle gear changeover valid
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)
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 outline
BA24	Turret width
BA25	Turret reference position X
BA26	Turret reference position Z

Address (bit)	Outline
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
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 ot real Y

Address (bit)	Outline
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
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	—
BA73	Barrier valid/invalid 1 (chuck, sub-chuck, tailstock)
BA74	Barrier valid/invalid 2 (Lower turret, work 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	Tail barrier reference position
BA91	Distance from spindle edge to partition plate
BA92	Central position X when viewed from machine zero point

Address (bit)	Outline
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)
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 W-axis thrust hold
BA106	Dead zone
BA107	Filter
BA108	W-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 - BA124	—
BA125 0	EIA tailstock barrier, tailstock usage valid/invalid
1	Tailstock present/absent

Address (bit)		Outline
	2	EIA tool barrier, tool holder present/absent
	3	Barrier tool nose position (EIA, automatic)
	4	Setting of BA11 is handled as tailstock extruding length, valid/invalid
	5	Output of spindle revolution M-code specified in same block as synchronous tapping, valid/invalid
	6	—
	7	—
BA126	0	System to be made valid without system selection for queuing
	1	Y-axis interference type
	2	Y-axis moving range display
	3	Tool command scheme
	4	FLASH tool valid/invalid
	5	Milling spindle orientation command (M219), output/no output
	6	—
	7	—
BA127 - BA132		—

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	—
TAP8	—
TAP9	“[” code for paper tape reader/puncher for EIA
TAP10	“]” code for paper tape reader/puncher for EIA
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
TAP17 - TAP24	—
TAP25	Paper tape puncher parity-V check
TAP26	Bit parameter related to paper tape reader/puncher

Address (bit)	Outline
TAP27	Bit parameter related to program end code (M) for paper tape reader
TAP28	—
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
TAP32	—

3. DNC parameter (DNC)

Address (bit)	Outline
DNC1	Type of terminator
DNC2	Terminator code 1
DNC3	Terminator code 2
DNC4	—
DNC5	DC code parity
DNC6 - DNC8	—
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)

Address (bit)		Outline
DNC22 - DNC24		—
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
	1	—
	2	Handling of tool data and tool files in the M PLUS format, valid/invalid
	3	—
	4	—
	5	—
	6	—
	7	—
DNC26	0	After program reception, a search is made/not made for the work number of that program.
	1	Details of an alarm occurring in DNC are displayed or not.
	2	Loading of programs having the same work number as that of the registered program in NC becomes impossible or not.
	3	The function of the PROGRAM LOCK/ ENABLE switch is released or not.
	4	—
	5	Three digit G-format and G10 format codes input/output for MAZAK data transfer protocol
	6	Binary to ASCII format input/output of MAZAK data transfer protocol
	7	All programs having work numbers smaller than No. 9000 are erased/not erased at the start of program reception.
DNC27		—
DNC28		—
DNC29		Number of retry times with detection of a physical error
DNC30		Tool data/tool file message format
DNC31		—
DNC32		—

4. Other (IOP/DPR/IDD)

Address (bit)	Outline
IOP1 - IOP4	—
IOP5	Loading a program(s) of the same work number, alarm without overwriting/overwriting
IOP6	—
IOP7	Data entry for communication with the magazine-side display unit
IOP8	—
IOP9	Number of pitch error axes during text output
IOP10 - IOP16	—
DPR1	Baud rate
DPR2	Stop bit
DPR3	—
DPR4	Data bit
DPR5 - DPR7	—
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	—
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]	

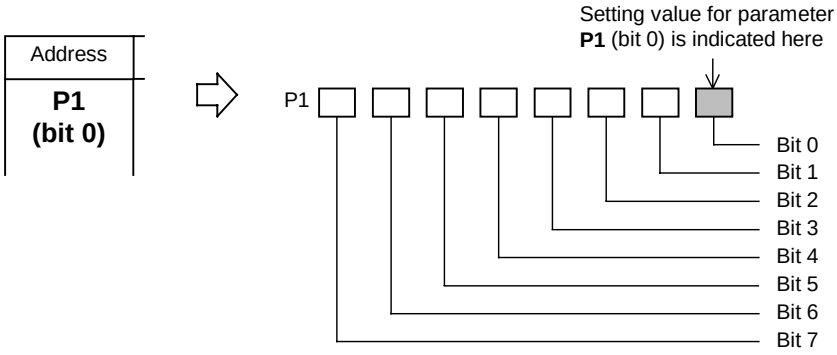
[1] Classification of parameters (USER, MACHINE or DATA I/O)

[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: In the parameter list, "At I/O startup" means that 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.

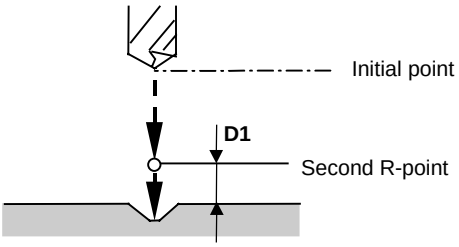
[7] Units of data displayed

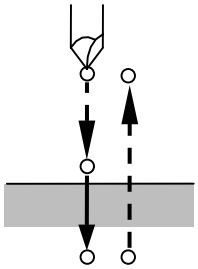
[8] Allowable range of data

[9] Details or meaning of the parameter

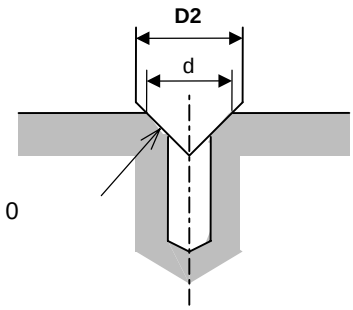
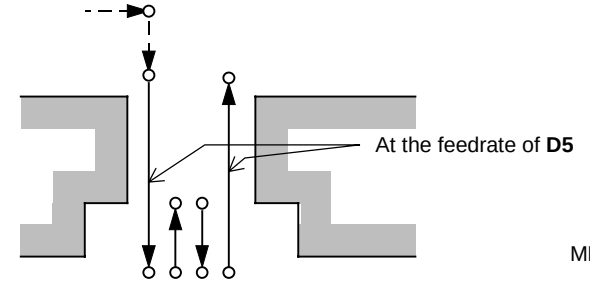
2-3-2 User parameter POINT (D)

Classification	USER	Display title	POINT

<table>
<tr><th>Address</th><th colspan="2">Meaning</th><th>Description</th></tr>
<tr><td rowspan="5">D1</td><td colspan="2">Height of the second R-point during point machining</td><td rowspan="5">Height of the second R-point
 

MPL001
The height of the R-point during point machining is basically D41, however, it is changed to D1 under the following conditions.
<table><tr><th>Tool sequence</th><th>Conditions</th></tr><tr><td>Drill</td><td>- 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>- 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></table>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)</td></tr>
<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 inch</td></tr>
<tr><td>Setting range</td><td>–999 to 999</td></tr>
<tr><td rowspan="5">D2</td><td colspan="2">Nominal diameter of spot-machining tool</td><td rowspan="5">The nominal diameter of a spot-machining tool that is automatically set during automatic tool development.
Example:
SNo. TOOL NOM-ϕ No. HOLE-ϕ HOLE-DEP
1 CTR-DR (20.) 10. ◆
↑
D2</td></tr>
<tr><td>Program type</td><td>M</td></tr>
<tr><td>Conditions</td><td>Immediate</td></tr>
<tr><td>Unit</td><td>1 mm/0.1 inch</td></tr>
<tr><td>Setting range</td><td>0 to 99</td></tr>
<tr><td rowspan="5">D3</td><td colspan="2">Number of revolutions during dwell at hole bottom in spot-machining cycle</td><td rowspan="5">Z-axis feed dwell time at the hole bottom in a spot-machining cycle. Set this time in spindle revolutions.
 
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.
3
MPL002</td></tr>
<tr><td>Program type</td><td>M</td></tr>
<tr><td>Conditions</td><td>Immediate</td></tr>
<tr><td>Unit</td><td>1 revolution</td></tr>
<tr><td>Setting range</td><td>0 to 9</td></tr>
</table>

Classification	USER	Display title	POINT

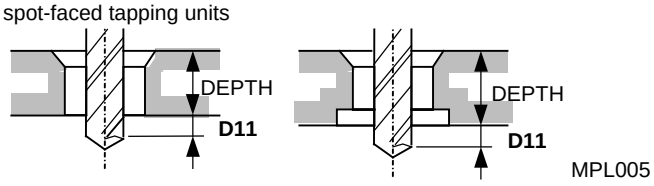
Address	Meaning		Description
D4	Maximum allowable spot-chamfering hole diameter element		Element used to set the maximum spot-chamfering hole diameter (d) during automatic tool development  Spot-chamfering occurs if d ≤ **D2** – **D4**. If d > **D2** – **D4**, the chamfering cutter is developed automatically. MPL003
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 mm/0.01 inch	
	Setting range	0 to 99	
D5	Prehole through speed during inversed spot-facing		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.  MPL004
	Program type	M	
	Conditions	Immediate	
	Unit	100 mm/min / 10 inch/min	
	Setting range	0 to 99	
D6 **D7**	Drill-machining cycle setting element		Element used to automatically set drill-machining cycles during automatic tool development
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 9	

Machining cycle	Conditions
Drilling cycle	$\frac{\textbf{DEPTH}}{\textbf{DIA}} \leq \textbf{D6}$
High-speed deep-hole drilling cycle	$\textbf{D6} < \frac{\textbf{DEPTH}}{\textbf{DIA}} \leq \textbf{D7}$
Deep-hole drilling cycle	$\textbf{D7} < \frac{\textbf{DEPTH}}{\textbf{DIA}}$

Classification	USER	Display title	POINT

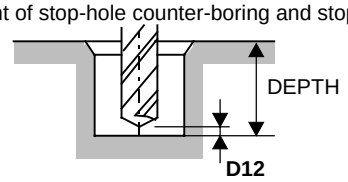
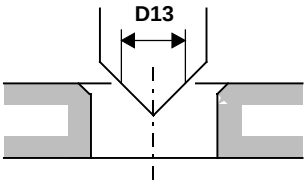
Address	Meaning		Description
D8	Maximum diameter of holes machinable on one drill		Element used to automatically set the number of drills which are automatically developed according to the hole diameter of the drill unit
	Program type	M	
	Conditions	Immediate	
	Unit	1 mm/0.1 inch	
	Setting range	0 to 99	
D9	Maximum diameter of holes machinable on two drills		
	Program type	M	
	Conditions	Immediate	
	Unit	1 mm/0.1 inch	
	Setting range	0 to 99	
D10	Maximum diameter of holes machinable on three drills		
	Program type	M	
	Conditions	Immediate	
	Unit	1 mm/0.1 inch	
	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 inversed spot-facing, tapping, back-boring, through-hole drilling, through-hole counter-boring, and spot-faced tapping units
	Program type	M	**Example:**
	Conditions	Immediate	SNo. TOOL NOM-ϕ No. HOLE-ϕ HOLE-DEP
	Unit	0.1 mm/0.01 inch	1 CTR-DR 10. 10. (21.) ← (DEPTH + **D11**)
	Setting range	0 to 99	**Note:** See also parameter **D30** for tapping units.

Number of drills developed	Conditions
1	**DIA** ≤ **D8**
2	**D8** < **DIA** ≤ **D9**
3	**D9** < **DIA** ≤ **D10**
Alarm	**D10** < **DIA**

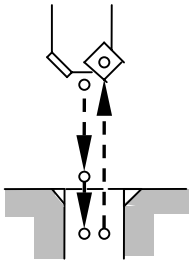
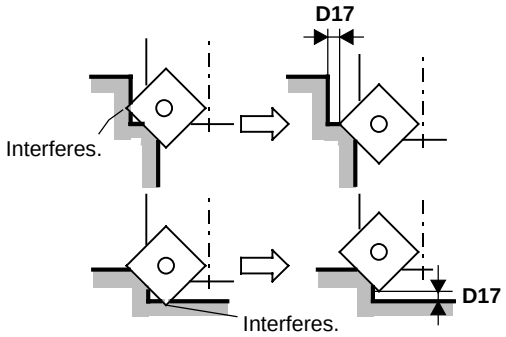
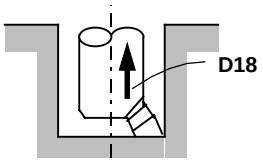
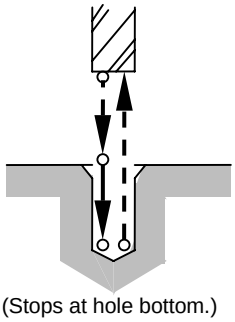


MPL005

Classification	USER	Display title	POINT

Address	Meaning		Description
D12	Stop-hole machining hole-bottom clearance		Element used to automatically set the hole-drilling depth during automatic tool development of stop-hole counter-boring and stop-hole boring units  MPL006 **Example:** SNo. TOOL NOM-ϕ No. HOLE-ϕ HOLE-DEP 1 DRILL 10. 10. 19. (**DEPTH** – tool tip compensation – **D12**) **Note:** This parameter is invalid when the residual hole diameter is not 0.
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 mm/0.01 inch	
	Setting range	0 to 99	
D13	Spot-machining hole diameter (fixed value)		Hole diameter is automatically set during automatic tool development when spot-chamfering is not to be performed.  MPL007 **Example:** SNo. TOOL NOM-ϕ No. HOLE-ϕ HOLE-DEP 1 CTR-DR 20. 10. ◆ **D13**
	Program type	M	
	Conditions	Immediate	
	Unit	1 mm/0.1 inch	
	Setting range	0 to 99	
D14	Depth-of-cut setting element for drilling (**ALMINUM**)		Element used to automatically set the depth-of-cut per drilling operation during automatic tool development **HOLE-ϕ × D14** : when the material of the stock workpiece is **AL** (aluminum) in article **MAT. 6** **HOLE-ϕ × D15** : when the material of the stock workpiece is other than **AL** in article **MAT. 6**
	Program type	M	
	Conditions	Immediate	
	Unit	0.1	
	Setting range	0 to 10	
D15	Depth-of-cut setting element for drilling (except **AL**)		
	Program type	M	
	Conditions	Immediate	
	Unit	0.1	
	Setting range	0 to 10	

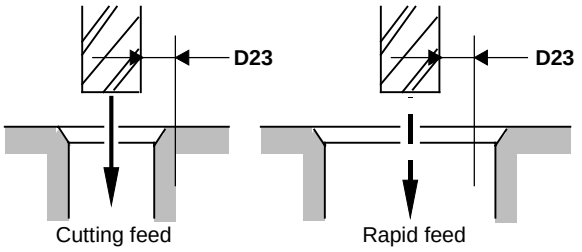
Classification	USER	Display title	POINT

Address	Meaning		Description
D16	Number of revolutions during dwell at hole bottom for chamfering cutter or spot-machining tool in chamfering cycle		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.  (Stops at hole bottom.) 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. **Note:** This parameter is invalid for chamfering with true-circle processing. MPL008
	Program type	M	
	Conditions	Immediate	
	Unit	1 revolution	
	Setting range	0 to 9	
D17	Interference clearance of chamfering cutter		The clearance in order to prevent tool interference with a wall of the workpiece or with the hole bottom during a chamfering cycle  MPL009
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 mm/0.01 inch	
	Setting range	0 to 99	
D18	Return feed rate for reaming or boring (cycle 3)		The feed rate at which the tool is returned from the hole bottom during reaming or boring.  MPL010 **Notes:** **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.
	Program type	M	
	Conditions	Immediate	
	Unit	100 mm/min / 10 inch/min	
	Setting range	0 to 9	
D19	Number of revolutions during dwell at hole bottom for end milling		Z-axis feed dwell time at the hole bottom in an end milling cycle. Set this time in spindle revolutions.  (Stops at hole bottom.) 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. **Note:** This parameter is invalid for true-circle processing. MPL011
	Program type	M	
	Conditions	Immediate	
	Unit	1 revolution	
	Setting range	0 to 999	

Classification	USER	Display title	POINT

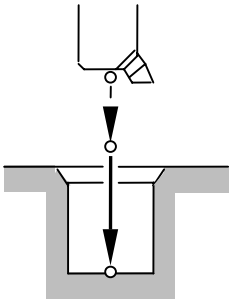
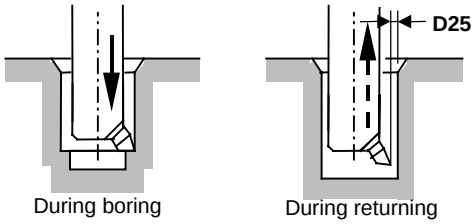
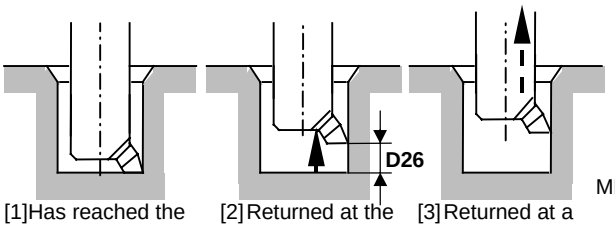
Address	Meaning		Description
D20	Radial depth-of-cut setting element for end milling		Element used to automatically set the radial depth-of-cut per end milling operation Depth-of-cut = nominal diameter × **D20** Depth-of-cut is automatically set according to the value of this parameter when nominal diameter of the end mill is input. **Example:** 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** × **D20**)
	Program type	M	
	Conditions	Immediate	
	Unit	1%	
	Setting range	0 to 100	
D21	Reference bottom-finishing allowance for end milling		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.
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 mm/0.01 inch	
	Setting range	0 to 99	
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**. **Note:** This parameter is valid only when the setting for roughness (**RGH**) of tapping (tool sequence) is **FIX**.
	Program type	M	
	Conditions	Immediate	
	Unit	0.01 sec.	
	Setting range	0 to 99	
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	1 mm/0.1 inch	
	Setting range	0 to 999	

Roughness	Bottom-finishing allowance
0 to 3	0.0
4	**D21**
5	**D21** × 0.7
6	**D21** × 0.7 × 0.7
7	**D21** × 0.7 × 0.7 × 0.7
8	**D21** × 0.7 × 0.7 × 0.7 × 0.7
9	**D21** × 0.7 × 0.7 × 0.7 × 0.7 × 0.7

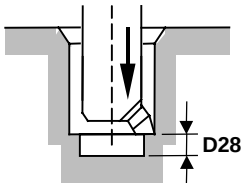
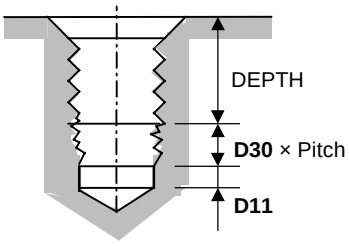


MPL012

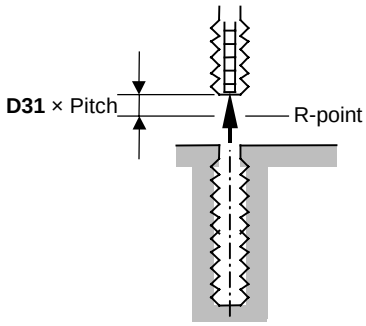
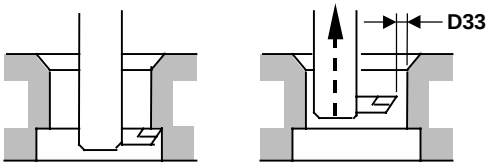
Classification	USER	Display title	POINT

Address	Meaning		Description
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. MPL013
	Program type	M	
	Conditions	Immediate	
	Unit	1 revolution	
	Setting range	0 to 9	
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  During boring During returning MPL014 **Notes:** **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**.
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 mm/0.01 inch	
	Setting range	0 to 99	
D26	Returning distance from hole bottom for boring or back-boring		The distance which the boring or back-boring tool is returned at the same feed rate as for cutting after the tool has reached the hole bottom  [1]Has reached the hole bottom. [2] Returned at the same feedrate. [3] Returned at a rapid feedrate. MPL015 **Note:** Not valid if the setting for the roughness (**RGH**) of the boring (tool sequence) is 1.
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 mm/0.01 inch	
	Setting range	0 to 99	
D27	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

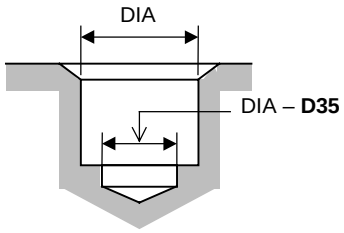
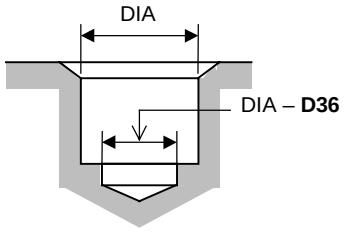
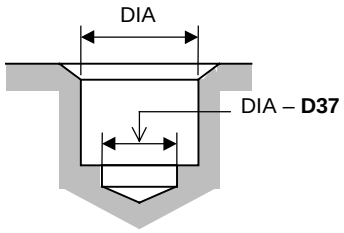
Classification	USER	Display title	POINT

Address	Meaning		Description
D28	Bottom-finishing amount of boring		The distance which the boring bar is fed in at 70% of the original feed rate to finish the hole bottom  MPL016 The feed rate is reduced to 70% of the original value before the hole bottom is reached. **Note:** Not valid if the setting for the roughness (**RGH**) of the boring (tool sequence) is 1.
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 mm/0.01 inch	
	Setting range	0 to 99	
D29	Chip removal time		The time required for a chip removal tool to complete a chip removal operation after the tool has been positioned to the hole
	Program type	M	
	Conditions	Immediate	
	Unit	1 sec.	
	Setting range	0 to 99	
D30	Number of incomplete threads in tapping cycle		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. This is also used as an element for automatically determining hole-drilling depth (**HOLE-DEP**) in the automatic tool development of the tapping unit.  MPL07 **Example:** SNo. TOOL NOM-ϕ No. HOLE-ϕ HOLE-DEP 1 DRILL 10. 10. 19. ↑ {**DEPTH** + **D11** + (**D30** × pitch)} [Related parameter] **D43:** Number of incomplete threads in tapping cycle for piped screws
	Program type	M	
	Conditions	Immediate	
	Unit	1 thread	
	Setting range	0 to 9	

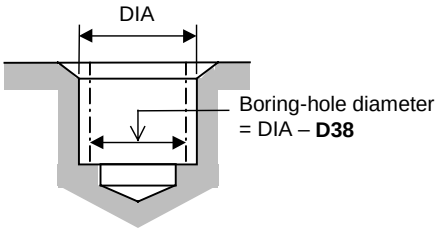
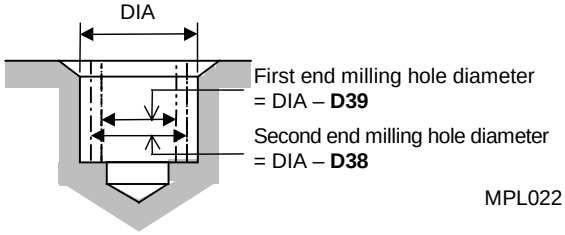
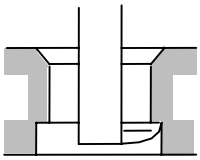
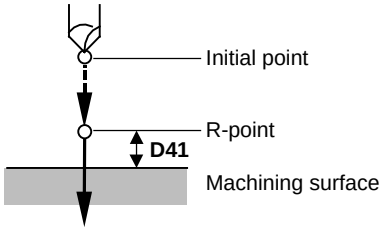
Classification	USER	Display title	POINT

Address	Meaning		Description
D31	Tapper elongation amount for tapping		Excess amount of tool return due to elongation of the tapper during tapping cycle Set this value in spindle revolutions.  MPL019
	Program type	M	
	Conditions	Immediate	
	Unit	1 revolution	
	Setting range	0 to 9	
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
	Program type	M	
	Conditions	Immediate	
	Unit	1 revolution	
	Setting range	0 to 99	
D33	Back-boring tool tip relief		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  [1] During back-boring [2] During passage MPL019 **Note:** For the relief direction of the tool tip, see the description of bit 3 and bit 4 of **I14**.
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 mm/0.01 inch	
	Setting range	0 to 99	
D34	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

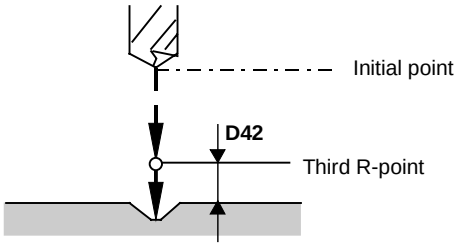
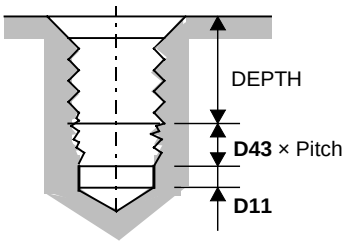
Classification	USER	Display title	POINT

Address	Meaning		Description
D35	Prehole-drilling diameter setting element for reamer (drilling)		Element used to automatically set the prehole-drilling diameter during automatic tool development of the reamer unit (When the pre-machining process is drilling.)  MPL020 **Example:** SNo. TOOL NOM-ϕ No. HOLE-ϕ 1 DRILL 10. (10.) ← (**DIA** – **D35**)
	Program type	M	
	Conditions	Immediate	
	Unit	0.01 mm/0.001 inch	
	Setting range	0 to 999	
D36	Prehole-drilling diameter setting element for reamer (boring)		Element used to automatically set the prehole-drilling diameter during automatic tool development of the reamer unit (When the pre-machining process is boring.)  MPL020 **Example:** SNo. TOOL NOM-ϕ No. HOLE-ϕ 1 DRILL 10. (10.) ← (**DIA** – **D36**)
	Program type	M	
	Conditions	Immediate	
	Unit	0.01 mm/0.001 inch	
	Setting range	0 to 999	
D37	Prehole-drilling diameter setting element for reamer (end milling)		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.)  MPL020 **Example:** SNo. TOOL NOM-ϕ No. HOLE-ϕ 1 DRILL 10. (10.) ← (**DIA** – **D37**)
	Program type	M	
	Conditions	Immediate	
	Unit	0.01 mm/0.001 inch	
	Setting range	0 to 999	

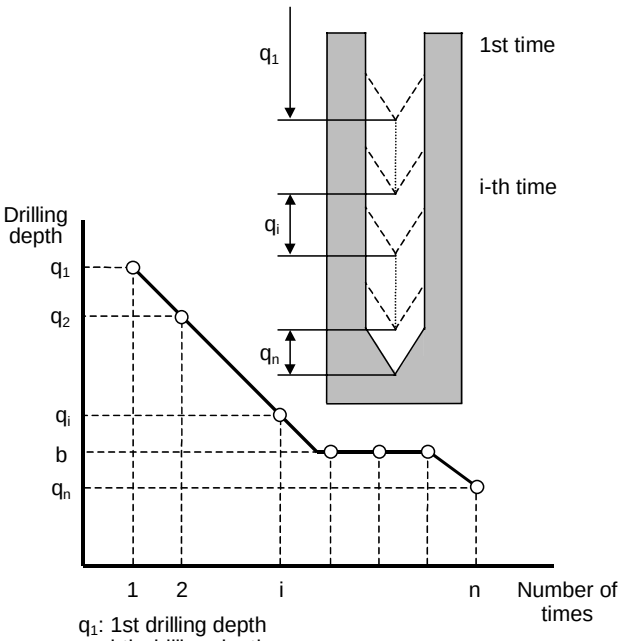
Classification	USER	Display title	POINT

Address	Meaning		Description
D38	Reamer-prehole diameter setting element for boring or end milling		1) In automatic tool development of the reamer unit, if the pre-machining process is boring:  MPL021 **Example:** SNo. TOOL NOM-ϕ No. HOLE-ϕ 1 BOR BAR 10. (10.) ← (DIA – **D38**)
	Program type	M	
	Conditions	Immediate	
	Unit	0.01 mm/0.001 inch	
	Setting range	0 to 999	
D39	Reamer-prehole diameter setting element for end milling		2) In automatic tool development of the reamer unit, if the pre-machining process is end milling:  MPL022 **Example:** SNo. TOOL NOM-ϕ No. HOLE-ϕ 1 END MILL 15. (20.) ← (DIA – **D39**) 2 END MILL 10. (21.) ← (DIA – **D38**)
	Program type	M	
	Conditions	Immediate	
	Unit	0.01 mm/0.001 inch	
	Setting range	0 to 999	
D40	Number of revolutions during dwell at spot-faced hole bottom for inversed spot-facing		Z-axis feed dwell time at the spot-faced hole bottom in an inversed spot facing cycle. Set this time in spindle revolutions.  (Feeding stops at hole bottom.) When the inversed 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. MPL023
	Program type	M	
	Conditions	Immediate	
	Unit	1 revolution	
	Setting range	0 to 9	
D41	R-point height during point-machining		R-point height of each tool in the point-machining unit **Example:**  MPL024 **Note:** For the inversed spot-facing unit or the back-boring unit, this parameter can also be used for setting the clearance amount at the hole bottom. (⇨**D1**, **D42**)
	Program type	M	
	Conditions	Immediate	
	Unit	1 mm/0.1 inch	
	Setting range	0 to 99	

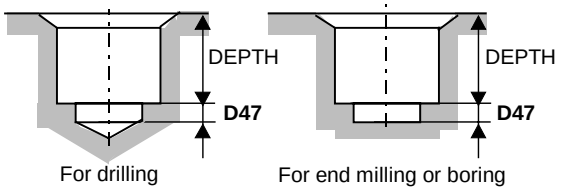
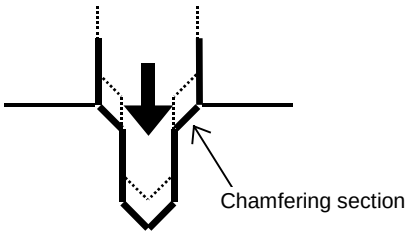
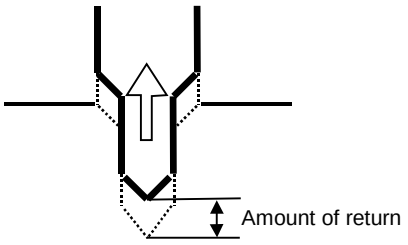
Classification	USER	Display title	POINT

Address	Meaning		Description
D42	Height of the third R-point during point machining		Height of the third R-point  MPL001 The height of the R-point during point machining is basically **D41**, however it is changed to **D42** under the following conditions. Tool sequence / Conditions: Drill: - 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: - Bit 7 of parameter **D91** is set to 1 (**D42** valid). - **CYCLE 2** is selected for the machining cycle. Spot: - Bit 7 of parameter **D91** is set to 1 (**D42** valid). - **CYCLE 2** in the chamfering cycle is selected for the machining cycle.
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 mm/0.01 inch	
	Setting range	–999 to 999	
D43	Number of incomplete threads in tapping cycle for piped screw		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. This is also used as an element for automatically determining hole-drilling depth (**HOLE-DEP**) in the automatic tool development of the tapping unit.  MPL07 **Example:** SNo. TOOL NOM-ϕ No. HOLE-ϕ HOLE-DEP 1 DRILL 10. 10. 19. ↑ {**DEPTH** + **D11** + (**D43** × pitch)}
	Program type	M	
	Conditions	Immediate	
	Unit	1 thread	
	Setting range	0 to 9	

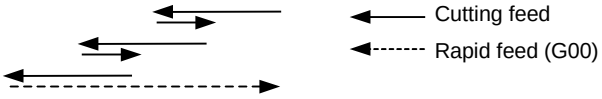
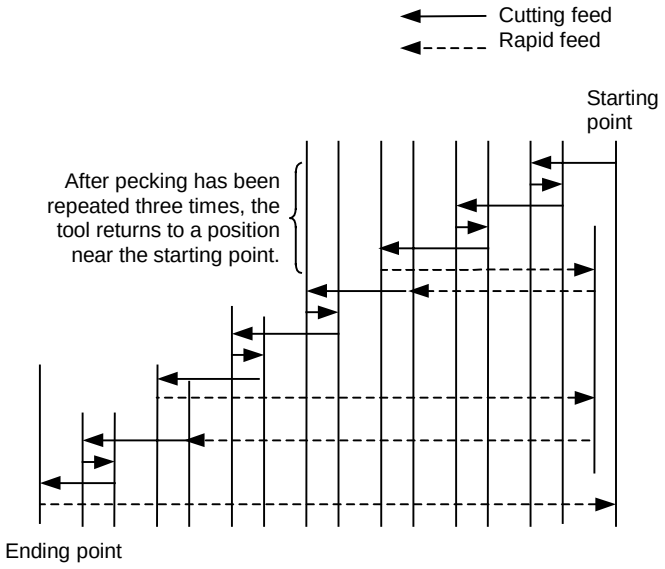
Classification	USER	Display title	POINT

Address	Meaning		Description
D44	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
D45	Gradual decrements in drilling depth		 q_1: 1st drilling depth q_i : i-th drilling depth q_n: Residual hole depth b : Minimum drilling depth **D46** $q_{i=} q_1 –$ **D45** × (i – 1) (If $q_i \geq$ b) $q_{i=}$ b (If $q_i <$ b)
	Program type	M	
	Conditions	Immediate	
	Unit	0.01 mm/0.001 inch	
	Setting range	0 to 9999	(⇨**D46**)
D46	Minimum gradual drilling depth		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.
	Program type	M	
	Conditions	Immediate	
	Unit	0.01 mm/0.001 inch	(⇨**D45**)
	Setting range	0 to 9999	

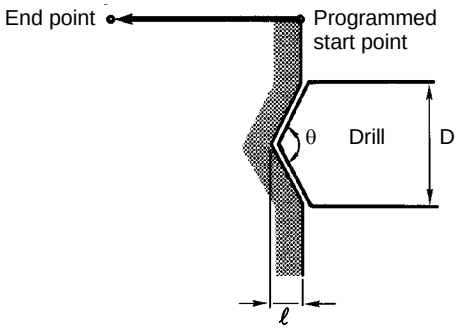
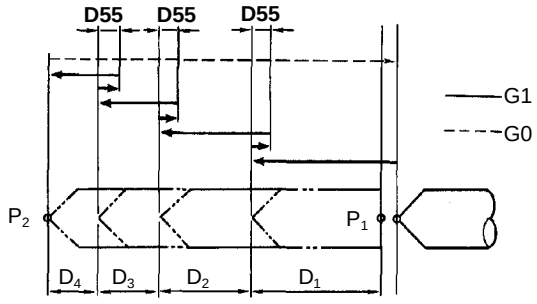
Classification	USER	Display title	POINT

Address	Meaning		Description
D47	Reamer-prehole machining overshoot		Element used to automatically set the hole depth (**HOLE-DEP**) of drilling, end milling and boring during automatic tool development of the reamer unit  For drilling For end milling or boring MPL025 **Example:** SNo. TOOL NOM-ϕ No. HOLE-ϕ HOLE-DEP 1 DRILL 10. 10. (21.) ← (DEPTH + **D47**)
	Program type	M	
	Conditions	Immediate	
	Unit	0.01 mm/0.001 inch	
	Setting range	0 to 999	
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 Chamfering feed = Pre-hole machining feed in tapping tool sequence × **D48**/100 
	Program type	M	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 999	
D49	Amount of return at hole bottom during the planetary tapping cycle		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 
	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	
	Conditions	Immediate	
	Unit	0.01 mm/rev 0.001 inch/rev	
	Setting range	0 to 999	

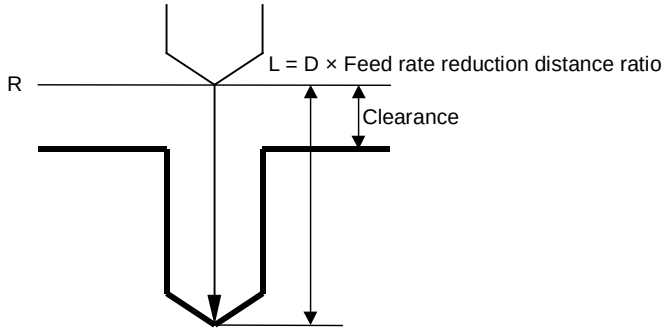
Classification	USER	Display title	POINT

Address	Meaning		Description
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 inch/rev	
	Setting range	0 to 999	
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	

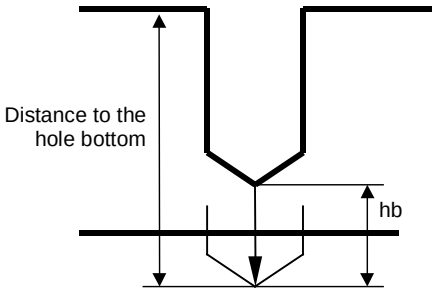
Classification	USER	Display title	POINT

Address	Meaning		Description
D54	Deceleration rate at cutting start for very-deep-hole drilling cycle/decremental very-deep-hole machining cycle		$F' = F \times \frac{D54}{100}$ F : Specified feed rate F' : Feed rate at cutting start  Feed rate at F' over distance ℓ from the programmed start point NM211-00268 $\begin{cases} \ell = \frac{D}{2 \times \tan\theta/2} & (0° < \theta < 180°) \\ \ell = 0 & (\theta \geq 180°) \end{cases}$
	Program type	M	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 100	
D55	Drilling return distance for very-deep-hole drilling cycle/decremental very-deep-hole machining cycle		 P_1: Start point P_2: End point D_n (n = 1 to 4) = Cut depth NM211-00252
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 9999	
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		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.
	Program type	M	
	Conditions	Immediate	
	Unit	Revoluitons	
	Setting range	0 to 255	

Classification	USER	Display title	POINT

Address	Meaning		Description
D57	Return speed for very-deep-hole drilling cycle/decremental very-deep-hole machining cycle		Set the return speed for the very-deep-hole drilling cycle or the decremental very-deep-hole machining cycle.
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/rev 0.0001 inch/rev	
	Setting range	0 to 9999	
D58	Feed rate reduction distance ratio at cutting start of a very-deep-hole drilling cycle (blind hole, through hole)		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 × **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). 
	Program type	M	
	Conditions	After movement stop	
	Unit	%	
	Setting range	0 to 300	

Classification	USER	Display title	POINT

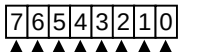
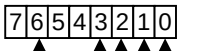
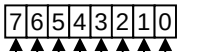
Address	Meaning		Description
D59	Circumferential speed reduction ratio at cutting end of a very-deep-hole drilling cycle (through hole)		Set the circumferential speed reduction ratio at cutting end of a very-deep-hole drilling cycle (through hole) during the point machining. If the circumferential speed during drilling is S, the circumferential speed S' at cutting end is calculated using the following calculation expression: S' = S × **D59**/100 The circumferential speed is reduced to S' by "speed reduction distance immediately before the hole bottom" (hb specified in the program). 
	Program type	M	
	Conditions	After movement stop	
	Unit	%	
	Setting range	0 to 100	
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 to **D72**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

Classification	USER

Display title	POINT

Address	Meaning		Description
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 For a **DEP-Z** range of the face mill, set a value of "**DEPTH**" (at a unit of 0.1 mm or 0.01 inch). 0 to **D76**..... **DEP-Z** range (for face mill) 1 **D76** to **D77** ..... **DEP-Z** range (for face mill) 2
	Program type	M	
	Conditions	Immediate	
	Unit	0.1% (0.1 mm/0.01 inch)	
	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 inch). 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 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
	Program type	M	
	Conditions	Immediate	
	Unit	0.1% (0.1 mm/0.01 inch)	
	Setting range	0 to 1000	
D83 to **D90**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

Classification	USER	Display title	POINT

Address	Meaning		Description
D91	—		 (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**. The R-point height of the chamfering cutter during the cycle 2 is set to **D42**. The R-point height of the spot-machining tool during the chamfering cycle (cycle 2) is set to **D42**.
	Program type	M	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	Binary, eight digits	
D92	—		 (1: Execution, 0: No execution) During a true-circle processing (end milling) cycle, **E17** is used for axial feed. The R1-point height of the back spot facing is set to **D1**. 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 to **D1**. 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**. During planetary tapping, chips are ejected automatically prior to the threading process.
	Program type	M	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	Binary, eight digits	
D93	—		Unidirectional positioning for point-machining  (1: Execution, 0: No execution) **CTR-DR** (Spot-machining tool) **DRILL** (Drill) **REAMER** (Reamer) **TAP** (Tap) **BCK FACE** (Inversed spot-facing tool) **BOR BAR** (Boring tool) **B-B BAR** (Back-boring tool) **CHAMFER** (Chamfering cutter)
	Program type	M	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	Binary, eight digits	

Classification	USER

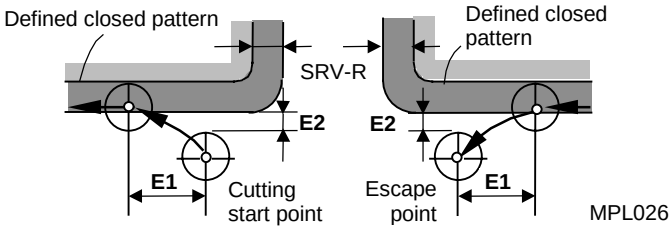
Display title	POINT

Address	Meaning		Description
D94	—		Unidirectional positioning for point-machining 7 6 5 4 3 2 1 0 (1: Execution, 0: No execution) **END MILL** (End mill) Planetary tapping Tornado cycle
	Program type	M	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	Binary, eight digits	
D95	Auto-setting method for tapping		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. 7 6 5 4 3 2 1 0 For pipe-tapping items 0: Conventional method 1: The text file concerned is referred to for the auto-setting of pipe-tapping data items in a MAZATROL program as well as on the **TOOL DATA** display. For unified thread items 0: Conventional method 1: The text file concerned is referred to for the auto-setting of the pre-hole diameter (**PRE-DIA**) within a tapping unit (for unified thread). For metric thread items 0: Conventional method 1: The text file concerned is referred to for the auto-setting of the pre-hole diameter (**PRE-DIA**) within a tapping unit (for metric thread).
	Program type	M	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	Binary, eight digits	
D96 to **D144**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

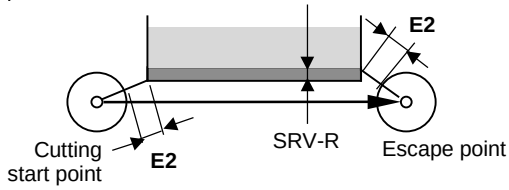
2-3-3 User parameter LINE/FACE/3D (E)

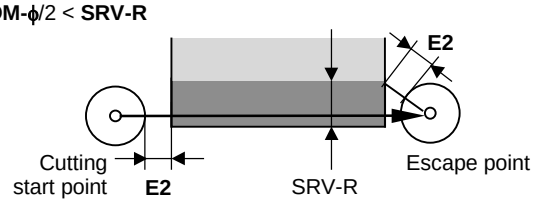
Classification	USER	Display title	LINE/FACE/3D

<table>
<tr><th>Address</th><th colspan="2">Meaning</th><th>Description</th></tr>
<tr><td rowspan="5">E1</td><td colspan="2">Closed-pattern cutting start point and escape point setting element</td><td rowspan="5">Element used to set cutting start point and escape point for closed-pattern line- or face-machining

Example:
 

MPL026

[Applicable units]
- LINE OUT, LINE IN, CHMF OUT and CHMF IN
- Wall finishing of STEP, POCKET, PCKT MT and PCKT VLY</td></tr>
<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 inch</td></tr>
<tr><td>Setting range</td><td>0 to 999</td></tr>
<tr><td rowspan="5">E2</td><td colspan="2">Cutting start point and escape point setting element (the first clearance)</td><td rowspan="5">Element used to set the cutting start point and escape point for line- or face-machining (the first clearance)
Example:
NOM-ϕ/2 ≥ SRV-R
 

NOM-ϕ/2 < SRV-R
 

MPL027

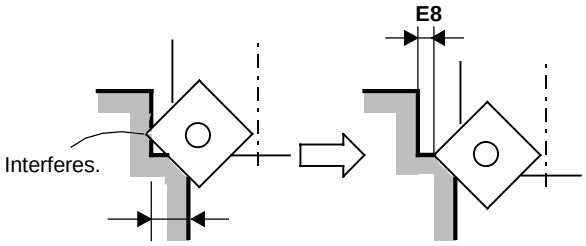
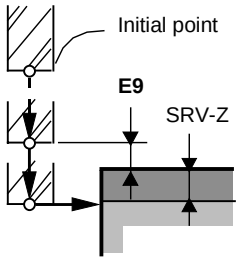
[Applicable units]
- All line-machining units
- Face-machining units other than FCE MILL, TOP EMIL, and SLOT

Notes:
1. See the diagram of parameter E1 also.
2. Positioning of E2 at the escape point can be selected using E95, but only for line-machining units.</td></tr>
<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 inch</td></tr>
<tr><td>Setting range</td><td>0 to 999</td></tr>
</table>

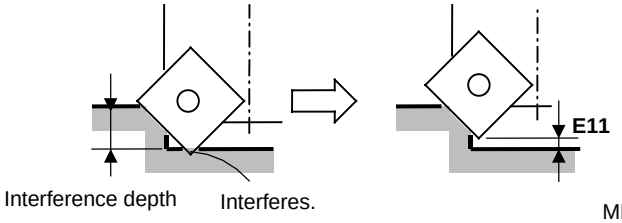
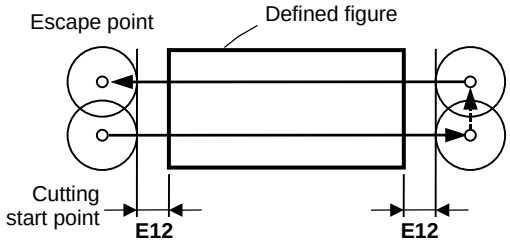
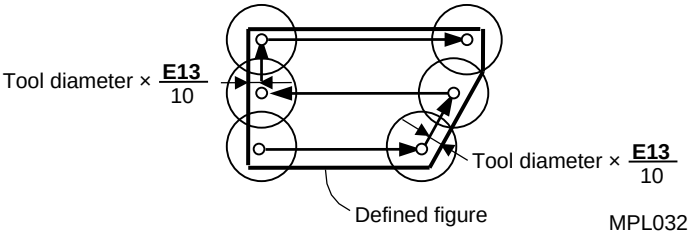
Classification	USER	Display title	LINE/FACE/3D

<table>
<tr><th>Address</th><th colspan="2">Meaning</th><th>Description</th></tr>
<tr><td rowspan="5">E3</td><td colspan="2">—</td><td>Invalid</td></tr>
<tr><td>Program type</td><td>—</td><td rowspan="4"></td></tr>
<tr><td>Conditions</td><td>—</td></tr>
<tr><td>Unit</td><td>—</td></tr>
<tr><td>Setting range</td><td>—</td></tr>
<tr><td rowspan="5">E4</td><td colspan="2">Reference allowance of finishing in radial direction</td><td rowspan="5">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.
<table><tr><th>Roughness</th><th>FIN-R</th></tr><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></table></td></tr>
<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 inch</td></tr>
<tr><td>Setting range</td><td>0 to 999</td></tr>
<tr><td rowspan="5">E5</td><td colspan="2">Element used to set the cutting start point and escape point (the second clearance)</td><td rowspan="5">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.</td></tr>
<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 inch</td></tr>
<tr><td>Setting range</td><td>0 to 999</td></tr>
<tr><td rowspan="5">E6</td><td colspan="2">Reference allowance of finishing in axial direction</td><td rowspan="5">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.
<table><tr><th>Roughness</th><th>FIN-Z</th></tr><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></table></td></tr>
<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 inch</td></tr>
<tr><td>Setting range</td><td>0 to 999</td></tr>
</table>

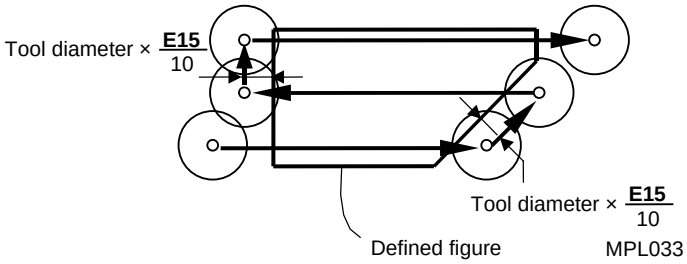
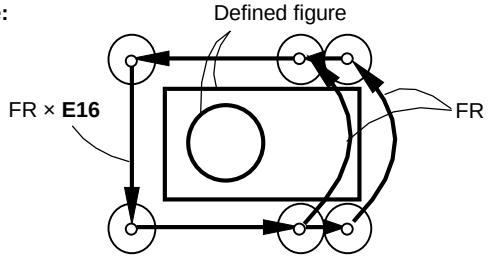
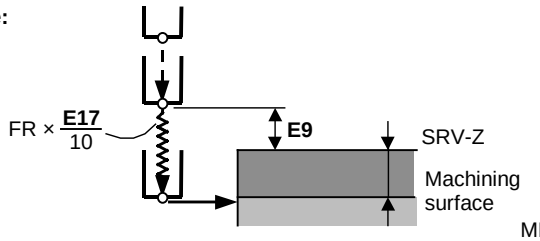
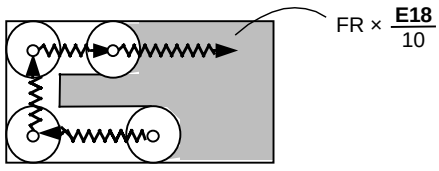
Classification	USER	Display title	LINE/FACE/3D

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 inch	
	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 inch	
	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 inch	
	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 **WID-R** = $\frac{\textbf{NOM-}\phi \times \textbf{E10}}{10}$ **Example:** SNo. TOOL NOM-ϕ No. APRCH-X APRCH-Y TYPE ZFD DEP-Z WID-R R1 FCE MILL 100A ? ? XBI ◆ 1. (19.) ↑ $\frac{\textbf{NOM-}\phi \times \textbf{E10}}{10}$
	Program type	M	
	Conditions	Immediate	
	Unit	10%	
	Setting range	0 to 9	

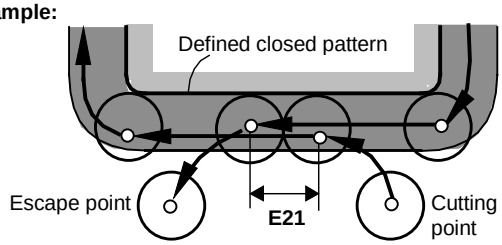
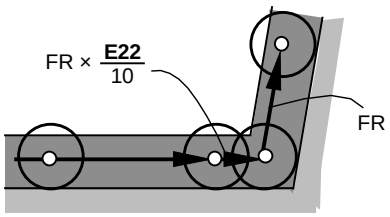
Classification	USER	Display title	LINE/FACE/3D

Address	Meaning		Description
E11	Axial interference clearance of chamfering cutter		The amount of clearance that prevents interference of the chamfering cutter with the bottom during chamfering  MPL030
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 mm/0.01 inch	
	Setting range	5 to 40	
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
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 mm/0.01 inch	
	Setting range	0 to 999	
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
	Program type	M	
	Conditions	Immediate	
	Unit	10%	
	Setting range	1 to 9	
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 **WID-R** = $\frac{\textbf{NOM-}\phi \times \textbf{E14}}{10}$ **Example:** SNo. TOOL NOM-ϕ No. APRCH-X APRCH-Y TYPE ZFD DEP-Z WID-R R1 END MILL 20. ? ? CW G01 10. (12.) ↑ $\frac{\textbf{NOM-}\phi \times \textbf{E14}}{10}$
	Program type	M	
	Conditions	Immediate	
	Unit	10%	
	Setting range	0 to 9	

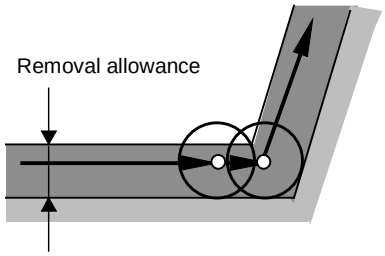
Classification	USER	Display title	LINE/FACE/3D

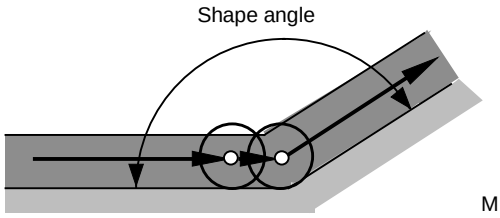
Address	Meaning		Description
E15	Tool path setting element for face milling-top unit (reciprocating short)		Element used to set the tool path external to the defined figure for reciprocating-short machining with face milling unit **Example:**  MPL033
	Program type	M	
	Conditions	Immediate	
	Unit	10%	
	Setting range	1 to 9	
E16	Peripheral-cutting feed rate override for end milling-mountain unit		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 **Note:** Valid only when bit 0 of **E91** is 1 and its bit 7 is 0. **Example:**  MPL034
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	1 to 20	
E17	Axial-cutting feed rate override		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 **Notes:** **1.** Valid only when **ZFD** of tool sequence is G01. **2.** Feed overriding is invalid when this parameter is 0. **Example:**  MPL035
	Program type	M	
	Conditions	Immediate	
	Unit	10%	
	Setting range	0 to 9	
E18	Override in case of the overall width cutting for pocket-machining		Override value of feed rate when the pocket-machining radial depth-of-cut becomes equal to the tool diameter **Example:**  MPL036
	Program type	M	**Note:** Overriding for overall width cutting is not valid when this parameter is 0. [Applicable units] Rough-machining of **POCKET**, **PCKT MT**, **PCKT VLY** and **STEP**
	Conditions	Immediate	
	Unit	10%	
	Setting range	0 to 9	

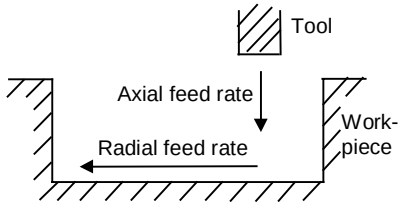
Classification	USER	Display title	LINE/FACE/3D

Address	Meaning		Description
E19	Returning feed rate override in case of bidirectional cutting for rough-machining of the end milling-slot unit.		Override value of tool returning feed rate in the rough-machining process of the end milling-slot unit, when the bidirectional cutting is executed. $\mathbf{FR} \times \frac{\mathbf{E19}}{100}$ **Note:** The override value is invalid, when bit 5 of parameter **E96** is set to OFF, or this parameter is set to 0. [Applicable unit] Returning path in rough-machining of **SLOT**
	Program type	M	
	Conditions	Immediate	
	Unit	1%	
	Setting range	0 to 999	
E20	Axial cutting feed override during Z-axial cutting in the pecking mode of face machining		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. Feed rate during pecking operation = $\mathbf{FR} \times \frac{\mathbf{E20}}{10}$ **Notes:** **1.** Valid only when **ZFD** in the tool sequence is G01. **2.** The feed override function is invalid when "0" is assigned to this parameter.
	Program type	M	
	Conditions	Immediate	
	Unit	10%	
	Setting range	0 to 9	
E21	Wall-cutting overlap in closed figure		The amount of overlap of the wall-cutting start and end areas in closed-pattern line- or face-machining **Example:**  MPL037 [Applicable units] - **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 inch	
	Setting range	0 to 999	
E22	Override value of automatic corner over-riding		Override value of automatic corner overriding in line- or face-machining **Example:**  MPL038 **Note:** Automatic corner overriding is invalid when this parameter is 0. [Applicable units] **LINE RGT**, **LINE LFT**, **LINE OUT**, **LINE IN**, **STEP**, **POCKET**, **PCKT MT** and **PCKT VLY**
	Program type	M	
	Conditions	Immediate	
	Unit	1%	
	Setting range	0 to 99	

Classification	USER	Display title	LINE/FACE/3D

<table>
<tr><th>Address</th><th colspan="2">Meaning</th><th>Description</th></tr>
<tr><td rowspan="5">E23</td><td colspan="2">Effective removal allowance (upper limit) of automatic corner overriding</td><td rowspan="9">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:
Tool diameter × E24/100 ≤ Removal allowance ≤ Tool diameter × E23/100
 

MPL039
<table><tr><th>Machining</th><th>Removal allowance</th></tr><tr><td>Line-rough machining</td><td>(SRV-R) – (FIN-R)</td></tr><tr><td>Face-rough machining</td><td>(WID-R)</td></tr></table></td></tr>
<tr><td>Program type</td><td>M</td></tr>
<tr><td>Conditions</td><td>Immediate</td></tr>
<tr><td>Unit</td><td>1%</td></tr>
<tr><td>Setting range</td><td>1 to 99</td></tr>
<tr><td rowspan="5">E24</td><td colspan="2">Effective removal allowance (lower limit) of automatic corner overriding</td></tr>
<tr><td>Program type</td><td>M</td></tr>
<tr><td>Conditions</td><td>Immediate</td></tr>
<tr><td>Unit</td><td>1%</td></tr>
<tr><td>Setting range</td><td>1 to 99</td></tr>
<tr><td rowspan="5">E25</td><td colspan="2">Effective angle (upper limit) of automatic corner overriding</td><td rowspan="5">The shape angle range (upper limit)
The automatic corner overriding becomes valid when the following line- or face-machining conditions are met:
Shape angle ≤ E25
 

MPL040</td></tr>
<tr><td>Program type</td><td>M</td></tr>
<tr><td>Conditions</td><td>Immediate</td></tr>
<tr><td>Unit</td><td>1°</td></tr>
<tr><td>Setting range</td><td>1 to 179</td></tr>
<tr><td rowspan="5">E26</td><td colspan="2">Calculation coefficient for the finishing feed of line milling</td><td rowspan="5">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.
Axial feed rate = Radial feed rate × E26/100
 
</td></tr>
<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 999</td></tr>
</table>

Classification	USER

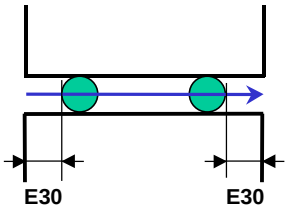
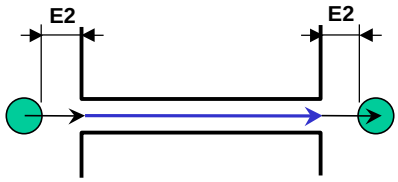
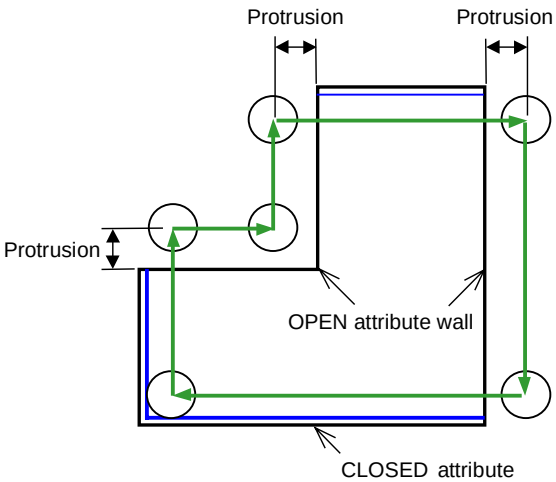
Display title	LINE/FACE/3D

<table>
<tr><th>Address</th><th colspan="2">Meaning</th><th>Description</th></tr>
<tr><td rowspan="5">E27</td><td colspan="2">Radial direction feed rate calculation reference diameter for finish cutting in line machining unit</td><td rowspan="5">These parameters, the roughness code, etc. determine the finishing feed rate.

- If the radial-finishing feed rate is taken as F_1, then:
$F_1 = \begin{cases} \textbf{E28} \times \dfrac{D}{\textbf{E27} \times \alpha} \times Kf \times Z & (D < \textbf{E27} \times \alpha) \\ \textbf{E28} \times Kf \times Z & (D \geq \textbf{E27} \times \alpha) \end{cases}$
D : Tool diameter
α : 0.1 (for metric specs.) or 0.01 (for inch specs.)
Kf : Roughness coefficient (Refer to the list below)
Z : Number of teeth

Roughness code / K_f:
∇ 1 — $K_0 \times 0.8^{-3}$
∇ 2 — $K_0 \times 0.8^{-2}$
$\nabla\nabla$ 3 — $K_0 \times 0.8^{-1}$
$\nabla\nabla$ 4 — K_0
$\nabla\nabla\nabla$ 5 — $K_0 \times 0.8$
$\nabla\nabla\nabla$ 6 — $K_0 \times 0.8^2$
$\nabla\nabla\nabla\nabla$ 7 — $K_0 \times 0.8^3$
$\nabla\nabla\nabla\nabla$ 8 — $K_0 \times 0.8^4$
$\nabla\nabla\nabla\nabla$ 9 — $K_0 \times 0.8^5$
K_0=Standard data 0.5</td></tr>
<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 inch</td></tr>
<tr><td>Setting range</td><td>0 to 65535</td></tr>
<tr><td rowspan="5">E28</td><td colspan="2">Finishing feed rate calculation reference feed rate in line machining unit</td></tr>
<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/rev
0.0001 inch/rev</td></tr>
<tr><td>Setting range</td><td>0 to 65535</td></tr>
<tr><td rowspan="5">E29</td><td colspan="2">Selection of whether the cutting conditions in the shape sequence during VFC mode are to be modified</td><td rowspan="5">Select whether the cutting conditions in the shape sequence during VFC mode are to be modified. The selection is valid only for milling.
0: Modification of the cutting conditions in the shape sequence
1: No modification of the cutting conditions in the shape sequence
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.</td></tr>
<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>

Classification	USER	Display title	LINE/FACE/3D

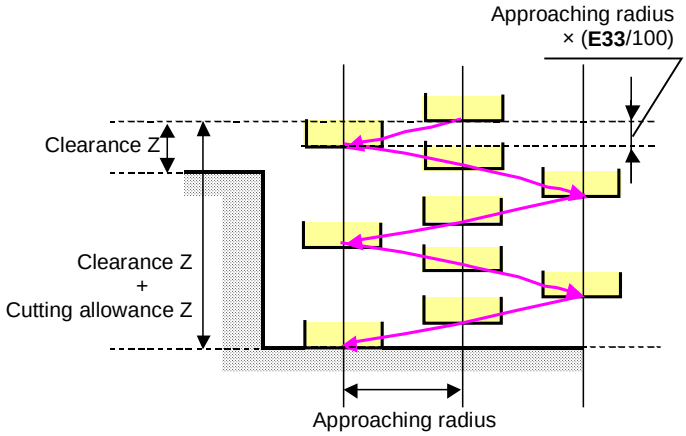
Address	Meaning		Description
E30	Radial clearance for wall attributes during line machining		An element that determines the starting point and escape point of radial cutting when **CLOSED** is specified for the wall attributes at the starting point and ending point of open-pattern line machining. When **OPEN** is specified, **E2** is used. <CLOSED specified>  <OPEN specified> 
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 mm/0.01 inch	
	Setting range	—	
E31	Element that determines the amount of **OPEN** attribute wall protrusion for pocket-machining shape units		Element that determines the amount of **OPEN** attribute wall protrusion in pocket-machining shape units (**POCKET**, **PCKT MT** or **PCKT VLY** unit only)  $\text{CLOSED attribute wall} = \frac{\text{Tool dia. (Note)} \times \textbf{E31}}{10}$ **Note:** Nominal tool diameter in the tool sequence is used when tool data is absent.
	Program type	M	
	Conditions	Immediate	
	Unit	10%	
	Setting range	0 to 9	

Classification	USER	Display title	LINE/FACE/3D

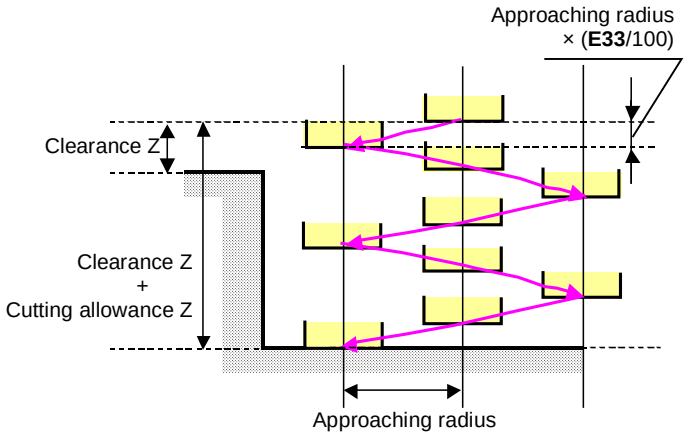
<table>
<tr><th>Address</th><th colspan="2">Meaning</th><th>Description</th></tr>
<tr><td rowspan="5">E32</td><td colspan="2" rowspan="2">Element that automatically determines an approaching radius in a Z-direction helical approach scheme</td><td rowspan="5">Element that automatically determines an approaching radius in a Z-direction helical approach scheme.

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:

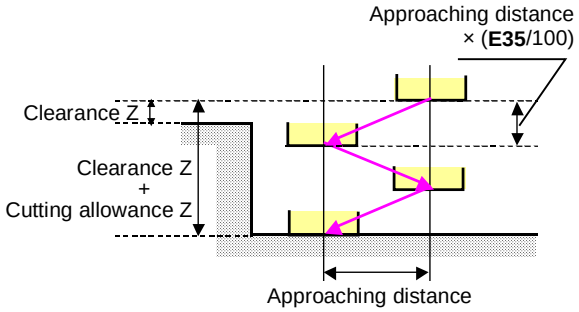
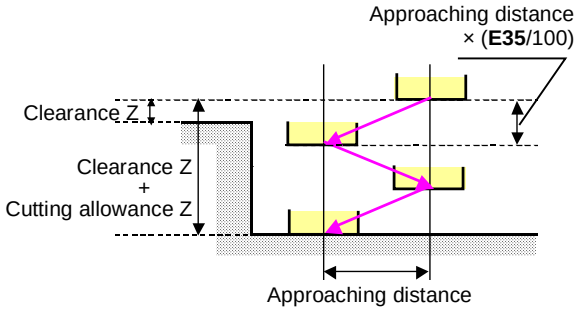
Approaching radius = $\frac{\text{Tool dia.} \times \textbf{E32}}{100}$

- For helical approach (Helical circle 2 + 1/4)
 
</td></tr>
<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>%</td></tr>
<tr><td>Setting range</td><td>1 to 999</td></tr>
<tr><td rowspan="5">E33</td><td colspan="2" rowspan="2">Approaching gradient during a helical approach scheme</td><td rowspan="5">Approaching gradient value during the helical approach scheme

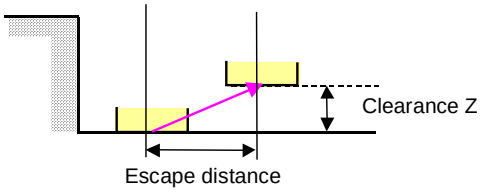
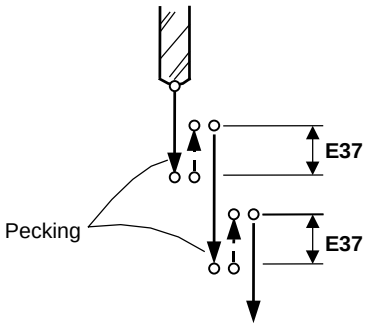
$\frac{\text{Distance in Z-direction}}{\text{Distance in XY-plane}}$

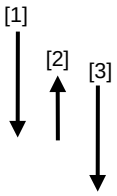
- For helical approach (Helical circle 2 + 1/4)
 
</td></tr>
<tr><td>Program type</td><td>M</td></tr>
<tr><td>Conditions</td><td>After stop of movement</td></tr>
<tr><td>Unit</td><td>%</td></tr>
<tr><td>Setting range</td><td>1 to 999</td></tr>
</table>

Classification	USER	Display title	LINE/FACE/3D

<table>
<tr><th>Address</th><th colspan="2">Meaning</th><th>Description</th></tr>
<tr><td rowspan="5">E34</td><td colspan="2">Element that automatically determines an approaching distance in a Z-direction tapered approach scheme</td><td rowspan="5">The approaching distance value to be used in the Z-direction tapered approach scheme for tapered machining is automatically determined by the parameter as follows:
Approaching distance = $\frac{\text{Tool dia.} \times \textbf{E34}}{100}$
- For tapered approach (3 tapers)
 
</td></tr>
<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>%</td></tr>
<tr><td>Setting range</td><td>1 to 999</td></tr>
<tr><td rowspan="5">E35</td><td colspan="2">Approaching gradient during the tapered approach scheme</td><td rowspan="5">Approaching gradient value during the tapered approach scheme
$\frac{\text{Distance in Z-direction}}{\text{Distance in XY-plane}}$
- For tapered approach (3 tapers)
 
</td></tr>
<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>10%</td></tr>
<tr><td>Setting range</td><td>1 to 999</td></tr>
</table>

Classification	USER	Display title	LINE/FACE/3D

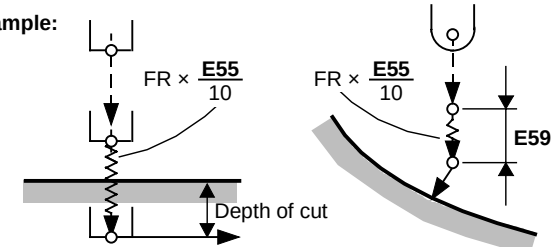
<table>
<tr><th>Address</th><th colspan="3">Meaning</th><th>Description</th></tr>
<tr><td>E36</td><td colspan="3">Element that automatically determines an escape distance in the Z-direction tapered escape scheme</td><td rowspan="5">The escape distance value to be used in the Z-direction tapered escape scheme after tapered machining is automatically determined by the parameter as follows:
Escape distance = $\frac{\text{Tool dia.} \times \textbf{E36}}{10}$
 
</td></tr>
<tr><td>Program type</td><td colspan="2">M</td></tr>
<tr><td>Conditions</td><td colspan="2">Immediate</td></tr>
<tr><td>Unit</td><td colspan="2">10%</td></tr>
<tr><td>Setting range</td><td colspan="2">0 to 10</td></tr>
<tr><td rowspan="6">E37</td><td colspan="3">Amount of return of pecking in the Z-axial pecking mode of face machining</td><td rowspan="6">This parameter indicates the amount of return of pecking in the Z-axial pecking mode of face machining.
 

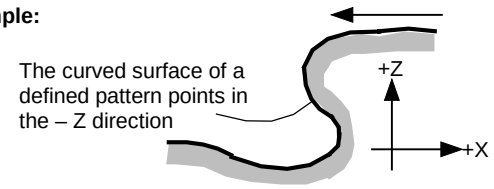
MPL058</td></tr>
<tr><td>Program type</td><td colspan="2">M</td></tr>
<tr><td>Conditions</td><td colspan="2">After stop of movement</td></tr>
<tr><td rowspan="3">Unit</td><td>Micron</td><td>0.001 mm
0.0001 inch</td></tr>
<tr><td>Submicron for rotational axis</td><td>0.001 mm
0.0001 inch</td></tr>
<tr><td>Submicron for all axes</td><td>0.0001 mm
0.00001 inch</td></tr>
<tr><td>Setting range</td><td colspan="2">0 to 99999999</td></tr>
<tr><td rowspan="5">E38</td><td colspan="3">Returning feed rate of pecking in the Z-axial pecking mode of face machining</td><td rowspan="5">Set the returning feed rate of the pecking in the Z-axial pecking mode during the face machining.
 

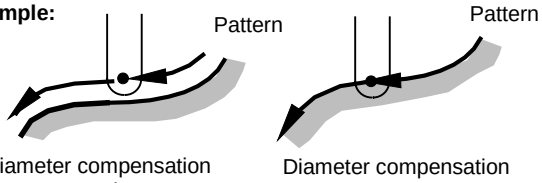
Feed rate
[1] Cutting (G1) F command
[2] Pecking (G1) E38
[3] Cutting (G1) F command

Note:
This value will be handled as100 if 0 is set here.</td></tr>
<tr><td>Program type</td><td colspan="2">M</td></tr>
<tr><td>Conditions</td><td colspan="2">Immediate</td></tr>
<tr><td>Unit</td><td colspan="2">0.01 mm/rev
0.001 inch/rev</td></tr>
<tr><td>Setting range</td><td colspan="2">0 to 9999</td></tr>
<tr><td rowspan="5">E39
to
E54</td><td colspan="3">—</td><td rowspan="5">Invalid</td></tr>
<tr><td>Program type</td><td colspan="2">—</td></tr>
<tr><td>Conditions</td><td colspan="2">—</td></tr>
<tr><td>Unit</td><td colspan="2">—</td></tr>
<tr><td>Setting range</td><td colspan="2">—</td></tr>
</table>

Classification	USER	Display title	LINE/FACE/3D

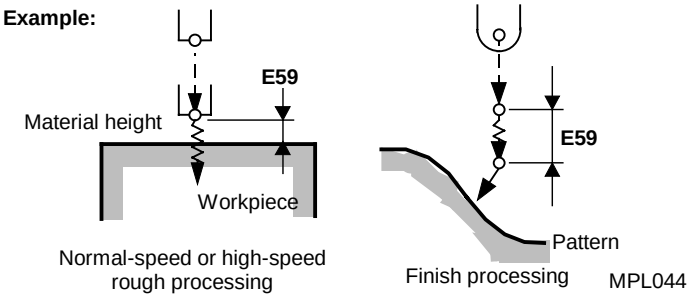
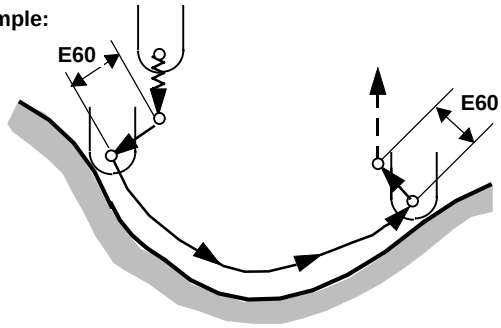
<table>
<tr><th>Address</th><th colspan="2">Meaning</th><th>Description</th></tr>
<tr><td rowspan="5">E55</td><td colspan="2">3-D
Axial cutting-feed overriding</td><td rowspan="5">Feed overriding for cutting a workpiece in an axial direction using a 3-D unit
Example:
 

MPL041
Note:
Feed overriding is invalid when this parameter is 0.</td></tr>
<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>
<tr><td rowspan="5">E56</td><td colspan="2">3-D
Inversion check of curved-surface pattern</td><td rowspan="5">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:
 

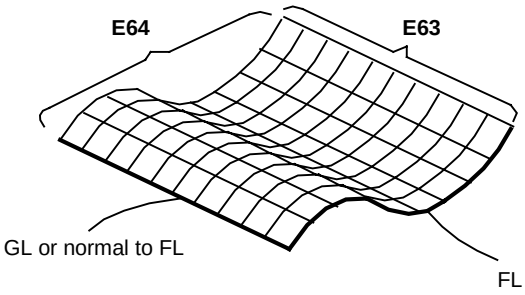
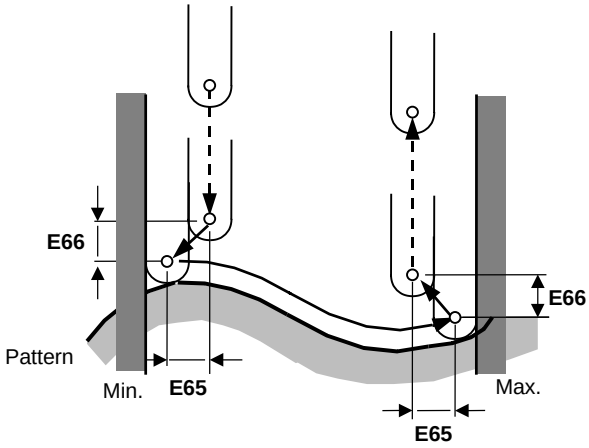
MPL042
Note:
This parameter is invalid during high-speed rough processing.</td></tr>
<tr><td>Program type</td><td>M</td></tr>
<tr><td>Conditions</td><td>Immediate</td></tr>
<tr><td>Unit</td><td>—</td></tr>
<tr><td>Setting range</td><td>0, 1</td></tr>
<tr><td rowspan="5">E57</td><td colspan="2">3-D
Severity check of cutting pitch</td><td rowspan="5">This parameter is used to select whether or not processing is to be performed in strict accordance with the tool-sequence pitch data setting.
0: The pitch setting is not strictly observed.
1: The pitch setting is strictly observed.
Notes:
1. This parameter is invalid during high-speed rough processing.
2. The operation time becomes long if this parameter is set to 1.</td></tr>
<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>
<tr><td rowspan="5">E58</td><td colspan="2">3-D
Tool-diameter compensation</td><td rowspan="5">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
0: Diameter compensation made
1: Diameter compensation not made
Example:
 

MPL043</td></tr>
<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>

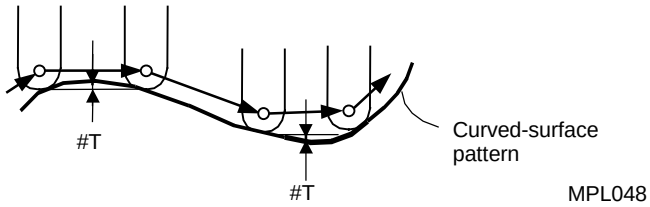
Classification	USER	Display title	LINE/FACE/3D

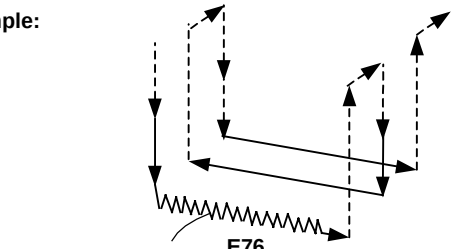
Address	Meaning		Description
E59	3-D Allowance of axial-cutting start position		Element used to set the position in which the cutting feed in axial direction is to be started after the tool has been moved from the initial point toward the workpiece at a rapid feed rate **Example:**  MPL044
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 mm/0.01 inch	
	Setting range	0 to 999	
E60	3-D Normal cutting allowance		The 3-D finish processing (cutting) allowance in the direction of the normal with respect to the defined pattern of the curved surface **Example:**  MPL045
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 mm/0.01 inch	
	Setting range	0 to 999	
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 inch) if 0 is set here.
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 mm/0.01 inch	
	Setting range	0 to 999	
E62	3-D Search length for right-angle cutting		
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 mm/0.01 inch	
	Setting range	0 to 999	

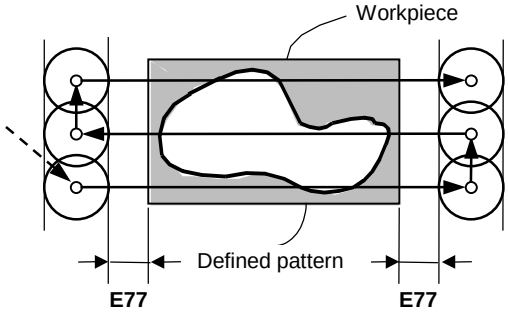
Classification	USER	Display title	LINE/FACE/3D

Address	Meaning		Description
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:**  MPL046 **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.
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 999	
E64	3-D Pattern display division segment (GL direction)		
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 999	
E65	3-D Radial cutting allowance for area check		The allowance of cutting a workpiece along the wall of the area which has been set using the area check function **Example:** 
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 mm/0.01 inch	
	Setting range	0 to 999	
E66	3-D Axial cutting allowance for area check		
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 mm/0.01 inch	
	Setting range	0 to 999	

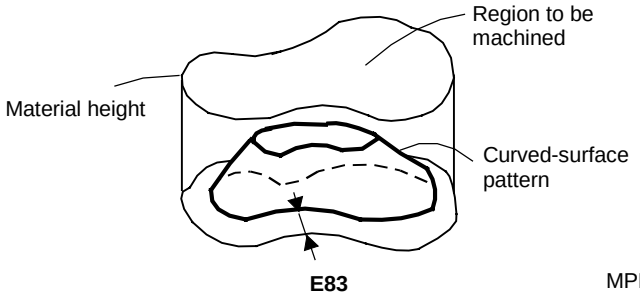
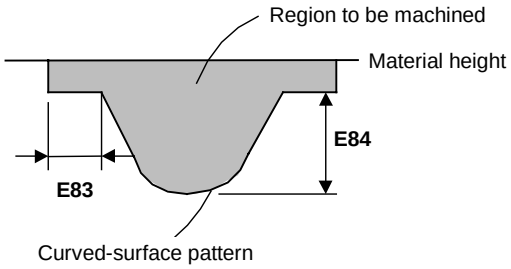
Classification	USER	Display title	LINE/FACE/3D

<table>
<tr><th>Address</th><th colspan="2">Meaning</th><th>Description</th></tr>
<tr><td rowspan="5">E67
to
E75</td><td colspan="2">3-D
Processing error tolerance</td><td rowspan="5">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><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>Example:
 

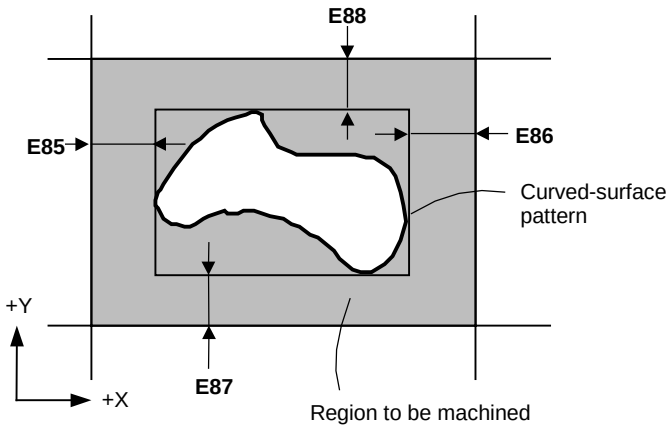
MPL048</td></tr>
<tr><td>Program type</td><td>M</td></tr>
<tr><td>Conditions</td><td>Immediate</td></tr>
<tr><td>Unit</td><td>0.01 mm/0.001 inch</td></tr>
<tr><td>Setting range</td><td>0 to 999</td></tr>
<tr><td rowspan="5">E76</td><td colspan="2">3-D
Entire-width override</td><td rowspan="5">The override 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:
 

MPL049
Note:
Entire-width overriding is not valid when this parameter is 0.</td></tr>
<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>
<tr><td rowspan="5">E77</td><td colspan="2">3-D
Radial cutting allowance for high-speed rough processing (workpiece size appointment)</td><td rowspan="5">The clearance of high-speed rough processing (workpiece size appointment) between the tool and the figure
 
</td></tr>
<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 inch</td></tr>
<tr><td>Setting range</td><td>0 to 999</td></tr>
<tr><td rowspan="5">E78</td><td colspan="2">3-D
Multiplying factor set for tolerance</td><td rowspan="5">0: 100%</td></tr>
<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>

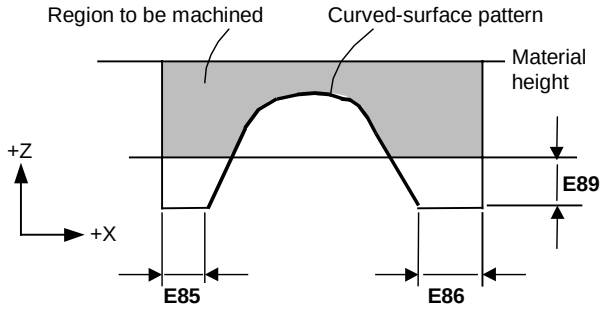
Classification	USER	Display title	LINE/FACE/3D

Address	Meaning		Description
E79 to **E82**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
E83	3-D Region of radial machining during high-speed rough processing (offset appointment)		That amount of offset from a curved-surface pattern which determines the region of high-speed rough processing (offset appointment) in a radial direction  MPL051
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 mm/0.01 inch	
	Setting range	0 to 9999	
E84	3-D Region of axial machining during high-speed rough processing (offset appointment)		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 **Example:**  MPL052
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 mm/0.01 inch	
	Setting range	0 to 9999	

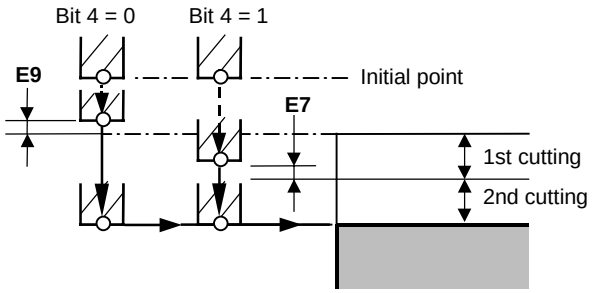
Classification	USER	Display title	LINE/FACE/3D

Address	Meaning		Description
E85	3-D Region of radial machining during high-speed rough processing: –X (workpiece size appointment)		The factor that determines the region of high-speed rough processing (workpiece size appointment) in a radial direction  MPL053
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 mm/0.01 inch	
	Setting range	0 to 9999	
E86	3-D Region of radial machining during high-speed rough processing: +X (workpiece size appointment)		
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 mm/0.01 inch	
	Setting range	0 to 9999	
E87	3-D Region of radial machining during high-speed rough processing: –Y (workpiece size appointment)		
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 mm/0.01 inch	
	Setting range	0 to 9999	
E88	3-D Region of radial machining during high-speed rough processing: +Y (workpiece size appointment)		
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 mm/0.01 inch	
	Setting range	0 to 9999	

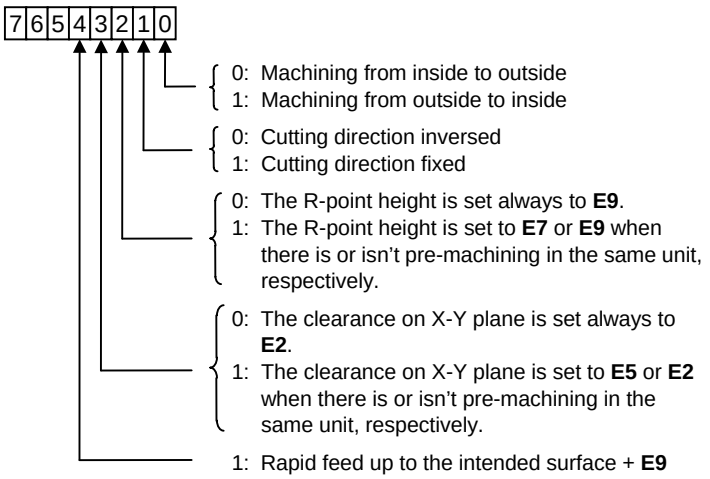
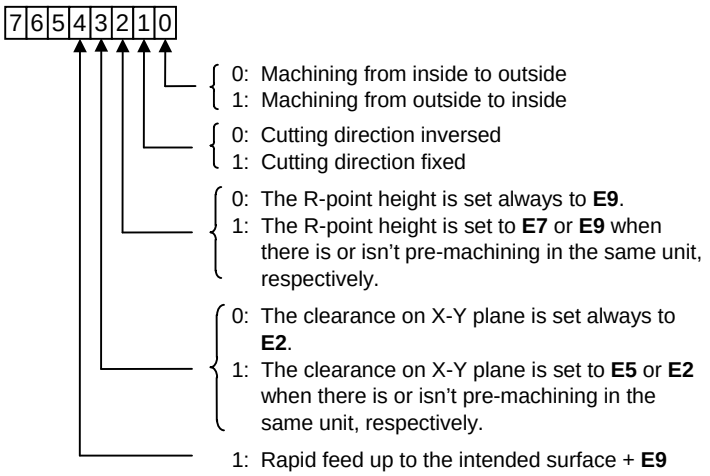
Classification	USER	Display title	LINE/FACE/3D

Address	Meaning		Description
E89	3-D Region of axial machining during high-speed rough processing (workpiece size appointment)		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 **Example:**  MPL054
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 mm/0.01 inch	
	Setting range	0 to 9999	
E90	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

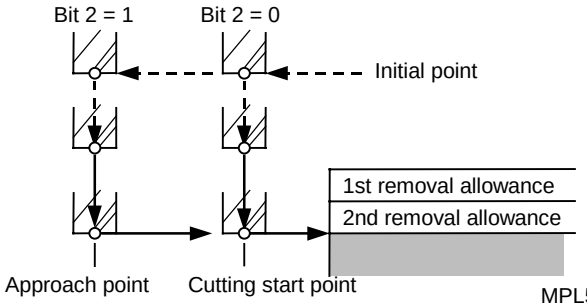
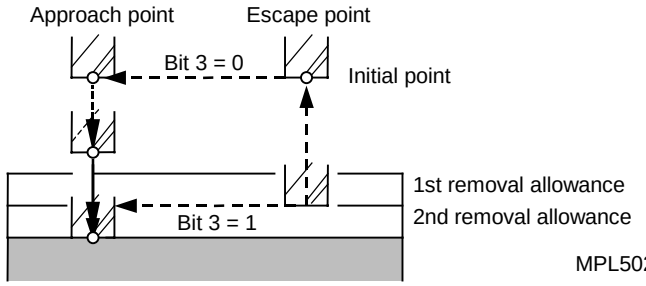
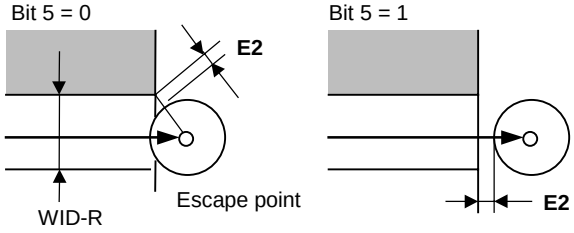
Classification	USER	Display title	LINE/FACE/3D

Address	Meaning	Description
E91	Tool-path pattern selection for end milling-mountain unit	7 6 5 4 3 2 4 0 0: Machining from inside to outside 1: Machining from outside to inside 0: Cutting direction inversed 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 **Notes:** **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	
E92	Tool-path pattern selection for pocket milling unit	7 6 5 4 3 2 1 0 0: Machining from inside to outside 1: Machining from outside to inside 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**
	Program type: M Conditions: Immediate Unit: Bit Setting range: Binary, eight digits	

Classification	USER	Display title	LINE/FACE/3D

<table>
<tr><th>Address</th><th colspan="2">Meaning</th><th>Description</th></tr>
<tr><td rowspan="5">E93</td><td colspan="2">Tool-path pattern selection for pocket milling-mountain unit</td><td rowspan="5"> 
</td></tr>
<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>
<tr><td rowspan="5">E94</td><td colspan="2">Tool-path pattern selection for pocket milling-valley unit</td><td rowspan="5"> 
</td></tr>
<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>

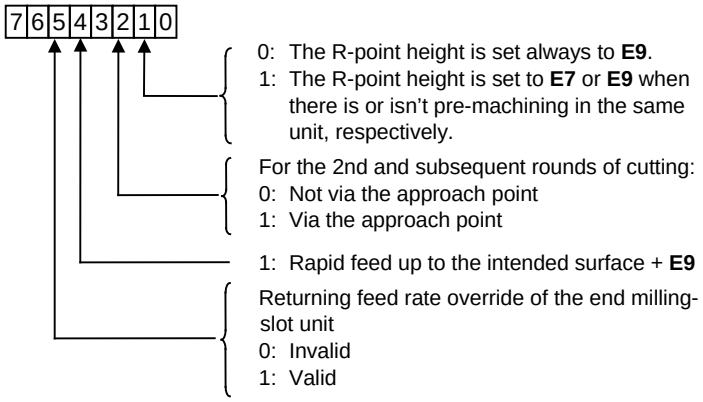
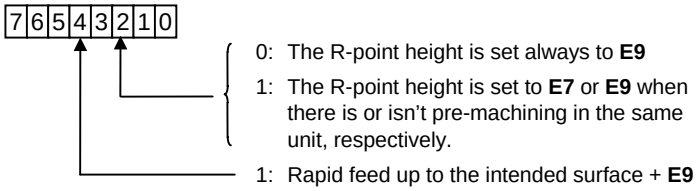
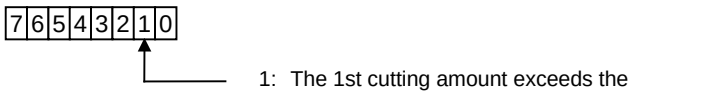
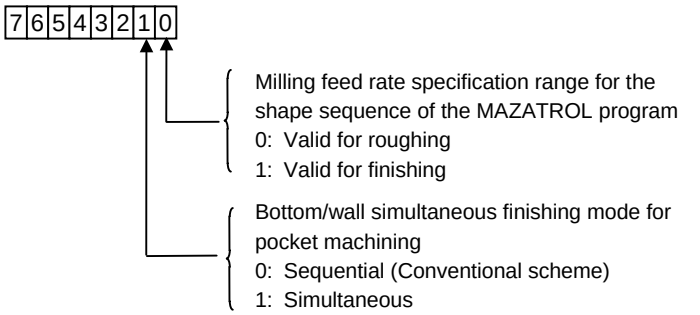
Classification	USER	Display title	LINE/FACE/3D

Address	Meaning	Description
E95	Tool-path pattern selection for line machining unit	7 6 5 4 3 2 1 0 For the 2nd and subsequent rounds of cutting: 0: Not via the approach point 1: Via the approach point For the 2nd and subsequent rounds of cutting: 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: 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: 0: Set always to **E2** 1: Set to **E5** or **E2** when there is or isn't pre-machining in the same unit, respectively. - Bit 2  MPL501 - Bit 3  MPL502 - Bit 5  MPL503 **Note:** Bit 3 valid only for inside/outside line machining unit.

Program type	M
Conditions	Immediate
Unit	Bit
Setting range	Binary, eight digits

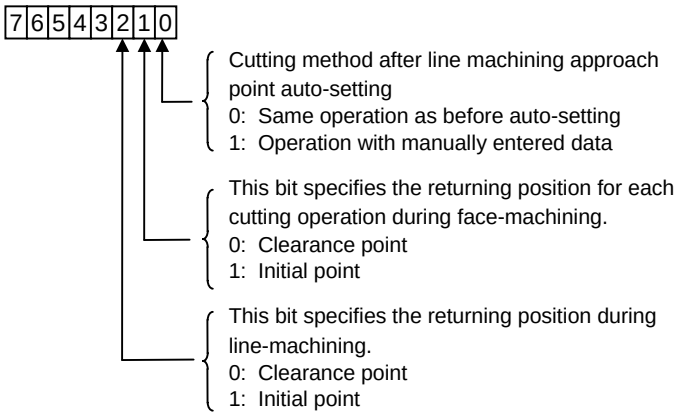
Classification	USER

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	
E97	Tool-path pattern selection for end milling-top unit		
	Program type	M	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	Binary, eight digits	
E98	Cutting method selection for end milling-mountain, pocket milling-valley unit		
	Program type	M	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	Binary, eight digits	
E99	—		
	Program type	M	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	Binary, eight digits	

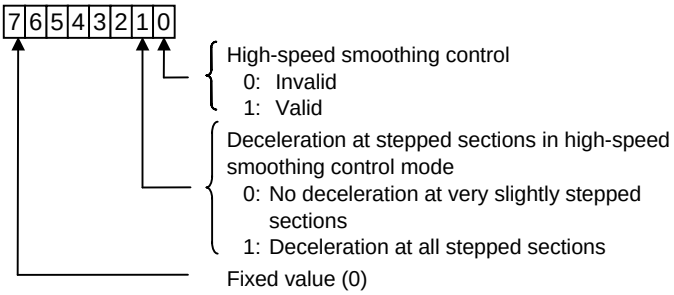
Classification	USER

Display title	LINE/FACE/3D

Address	Meaning		Description
E100 to **E103**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
E104	Tool path selection		
	Program type	M	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	Binary, eight digits	
E105 to **E144**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

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

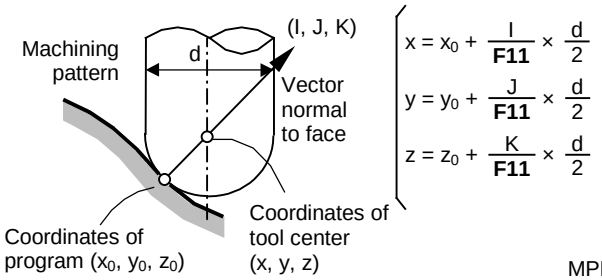
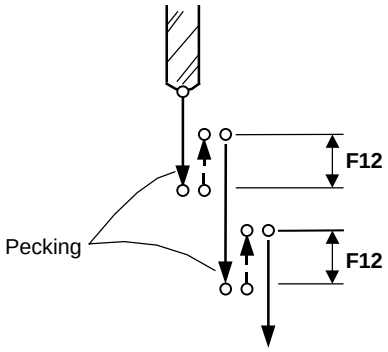
Classification	USER	Display title	EIA/ISO

Address	Meaning		Description
F1	Corner deceleration coefficient		When (**L74**/**L75**) × (**F1**/100) is assigned as the allowable acceleration attained before corner deceleration is started, the theoretical value Vo of the corner deceleration in G61.1 will be changed to the following Vo' value: Vo' = Vo × **F1**/100 **Note:** 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.
	Program type	M, E	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 500	
F2	Arc-clamping speed coefficient		When (**L74**/**L75**) × (**F2**/100) is assigned as the allowable acceleration attained before arc radius speed clamping is started, the theoretical value Vc of the arc radius clamping speed in G61.1 will be changed to the following Vc' value: $Vc' = Vc \times \sqrt{(\mathbf{F2}/100)}$ **Note:** 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.
	Program type	M, E	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 500	
F3	—		
	Program type	E	
	Conditions	After stop of movement	
	Unit	—	
	Setting range	0, 1	
F4	Fixed value (0)		
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

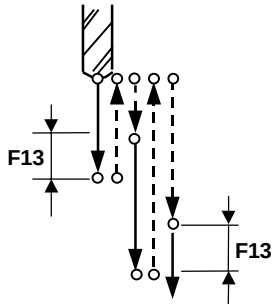
Classification	USER	Display title	EIA/ISO

Address	Meaning		Description
F5	Fixed value (0)		
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
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 inch	
	Setting range	0 to 100	
F7	Fixed value (0)		
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
F8	Corner deceleration speed coefficient for high-speed smoothing control		Corner deceleration speed Vo determined by the optimal corner deceleration function is adjusted using the value set in this parameter. $Vo = Vo \times \frac{\mathbf{F8}}{100}$ 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	

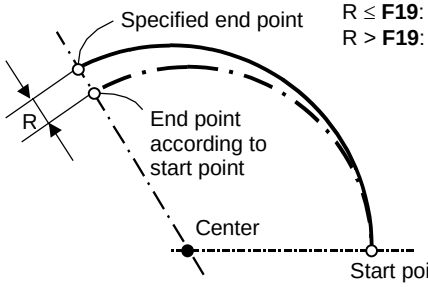
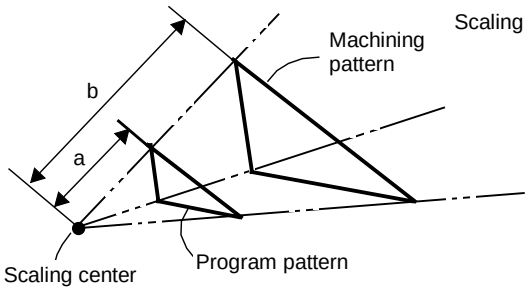
Classification	USER	Display title	EIA/ISO

Address	Meaning		Description
F9	Circler cutting clamp speed coefficient for high-speed smoothing control		Circler cutting clamp speed Vc is adjusted using the value set in this parameter. $Vc = Vc \times \sqrt{\frac{\mathbf{F9}}{100}}$ 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	—		Not used.
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
F11	Vector constant for 3-D tool-diameter compensation		 MPL057 **Note:** **F11** = $\sqrt{I^2+J^2+K^2}$ if this parameter is 0.
	Program type		
	Conditions	Next block	
	Unit	0.001 mm/0.0001 inch (0.001°)	
	Setting range	0 to 99999999	
F12	Return amount of pecking in drill high-speed deep-hole cycle or in G73		Return amount of pecking in drill high-speed deep-hole cycle or in G73 tool path  MPL058
	Program type	M, E	
	Conditions	Next block	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 99999999	

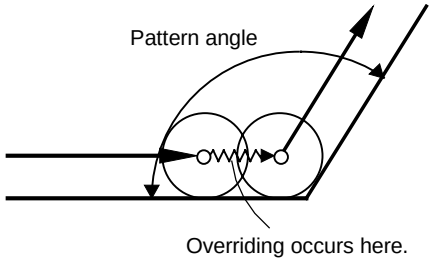
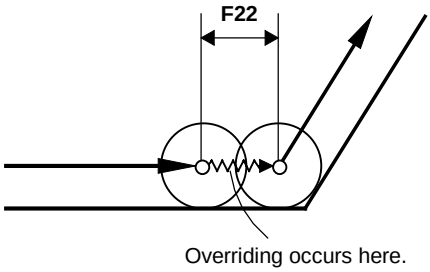
Classification	USER	Display title	EIA/ISO

Address	Meaning		Description
F13	Allowance amount of rapid-feed stop in deep-hole drilling cycle or in G83		The allowance amount provided for the tool to stop moving at rapid-feed just in front of the preceding hole during a deep-hole drilling cycle or during G83 tool path  MPL059
	Program type	M, E	
	Conditions	Next block	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 99999999	
F14	Rotation center of coordinates (axis of abscissa)		
	Program type	—	
	Conditions	At power on	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to ±99999999	
F15	Rotation center of coordinates (axis of ordinate)		
	Program type	—	
	Conditions	At power on	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to ±99999999	
F16	Horizontal length of coordinate rotation		Vector of coordinate rotation (axis of abscissa)
	Program type	—	
	Conditions	At power on	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to ±99999999	

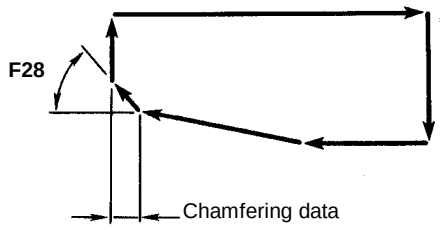
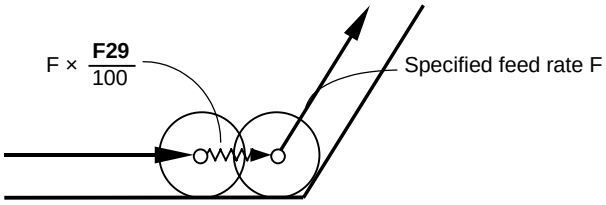
Classification	USER	Display title	EIA/ISO

Address	Meaning		Description
F17	Vertical length of coordinate rotation		Vector of coordinate rotation (axis of ordinate)
	Program type	—	
	Conditions	At power on	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to ±99999999	
F18	Angle of coordinate rotation		
	Program type	—	
	Conditions	At power on	
	Unit	0.001°	
	Setting range	0 to ±180000	
F19	Maximum permissible difference in arc radius		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. R ≤ **F19**: Spiral interpolation R > **F19**: Alarm  MPL060
	Program type	M, E	
	Conditions	Next block	
	Unit	0.001 mm/0.0001 inch (0.001°)	
	Setting range	0 to 9999	
F20	Fixed value of scaling factor		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. Scaling factor = $\frac{b}{a}$  MPL061
	Program type	E	
	Conditions	Next command	
	Unit	1/1000000	
	Setting range	0 to 99999999	

Classification	USER	Display title	EIA/ISO

Address	Meaning		Description
F21	Maximum inside-corner angle available with automatic corner override (G62)		The automatic corner override using the G62 code becomes valid when the following condition of the shape angle is met: Shape angle ≤ **F21**  MPL062
	Program type	E	
	Conditions	Next command	
	Unit	1°	
	Setting range	0 to 179	
F22	Deceleration area of automatic corner overriding (G62)		The area in which automatic corner overriding using the G62 code occurs  MPL063
	Program type	E	
	Conditions	Next command	
	Unit	0.001 mm/0.0001 inch (0.001°)	
	Setting range	0 to 99999999	
F23 to **F26**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
F27	Handling of G92 (spindle speed clamp value) command at restart		0: Only the last G92SsQqRr command is enabled. 1: All the G92SsQqRr commands are enabled. 2 to 8: Not used. 9: The spindel speed clamp value is invalid at restart.
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 9	

Classification	USER	Display title	EIA/ISO

Address	Meaning		Description
F28	Threading chamfering angle		Set chamfering angle at thread portion in thread cutting cycle. G76 and G92 are used for threading cutting cycle, if **F30** is set to 0. G276 and G292 are used for threading cutting cycle, if **F30** is set to 1.  NM211-00219 **Note:** Set 45 or 60.
	Program type	E	
	Conditions	Immediate	
	Unit	1°	
	Setting range	45, 60	
F29	Override value of automatic corner overriding (G62)		The override value of automatic corner overriding using the G62 code  MPL064 **Note:** The automatic corner overriding is invalid when this parameter is 0.
	Program type	E	
	Conditions	Next command	
	Unit	1%	
	Setting range	0 to 100	
F30	G-code type		Selecting 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.
	Program type	E	
	Conditions	At power on	
	Unit	Decimal number	
	Setting range	0, 1	
F31 to **F39**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

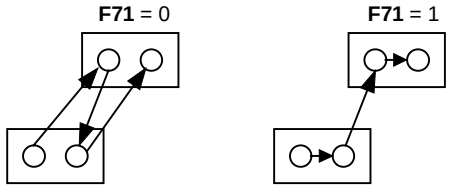
Classification	USER	Display title	EIA/ISO

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		Set a threading termination waiting time. **F41 =** 0, 1 or 128 to 255 No waiting time **F41** = 2 to 127 (Setting – 1) × 3.5 msec
	Program type	M	
	Conditions	Immediate	
	Unit	3.5 ms	
	Setting range	0 to 255	
F42	Deceleration area r during Z-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 Z_ Rr D_ F_;` (G37)
	Program type	E	
	Conditions	After stop of movement	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 99999999	
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 Z_ R_ Dd F_;` (G37)
	Program type	E	
	Conditions	After stop of movement	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 99999999	

Classification	USER	Display title	EIA/ISO

Address	Meaning		Description
F44	Measuring speed f		Measuring speed (f) This data is used when argument F is omitted in G37 command format. `G37 Z_ R_ D_ Ff;` Standard setting 1 to 60000 mm/min 1 to 2362 inch/min (G37)
	Program type	E	
	Conditions	After stop of movement	
	Unit	1 mm/min / 1 inch/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 X_ Rr D_ F_;` (G37)
	Program type	E	
	Conditions	After stop of movement	
	Unit	0.001 mm/0.0001 inch	
	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 X_ R_ Dd F_;` (G37)
	Program type	E	
	Conditions	After stop of movement	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 99999999	
F47 to **F66**	Common variable name		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
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

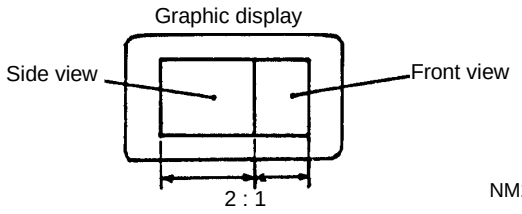
Classification	USER	Display title	EIA/ISO

Address	Meaning		Description
F67 **F68**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
F69	EIA/ISO program restart method		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.
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
F70	Availability of multiple-machining and designated number of repetitions in the EIA/ISO subprogram		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 effective for the EIA/ISO subprogram. Specified number of times to restart the EIA/ISO subprogram is effective. 1: Multiple-machining is ineffective for the EIA/ISO subprogram. Specified number of times to restart the EIA/ISO subprogram is ineffective.
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	0, 1	
F71	Machining order control		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 **F71** = 0 **F71** = 1  MPL065
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	

Classification	USER	Display title	EIA/ISO

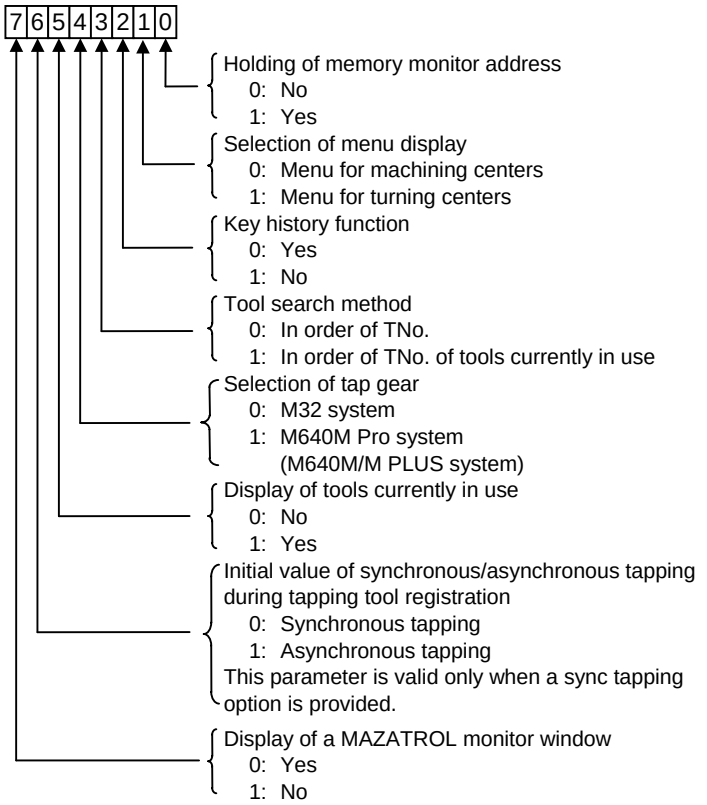
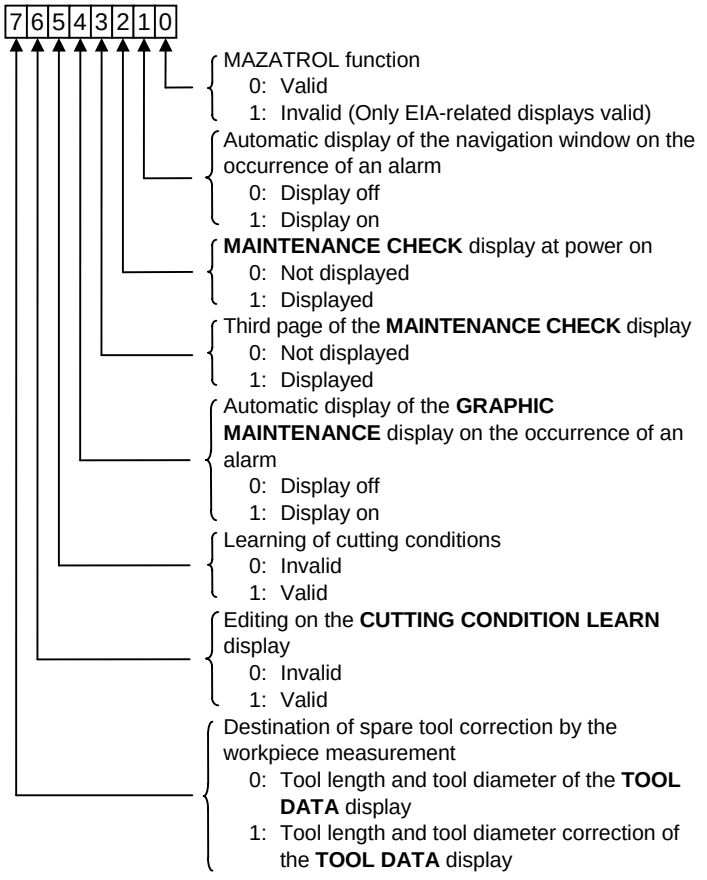
Address	Meaning		Description
F72	Selection of the shape correction function of the MAZATROL program		To select whether the shape correction function of the MAZATROL program is always effective or ineffective. 0: Invalid 1: Shape correction function only valid 2: Shape correction function and high-precision true-circle function (reversed type) valid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	0, 1, 2	
F73	M-code execution time for time study		The tool-path check time study time that is accumulated each time an M-code is output.
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.01 sec.	
	Setting range	0 to 10000	
F74	S-code execution time for time study		The tool-path check time study time that is accumulated each time an S-code is output.
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.01 sec.	
	Setting range	0 to 10000	
F75	T-code execution time for time study		The tool-path check time study time that is accumulated each time a T-code is output.
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.01 sec.	
	Setting range	0 to 10000	

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 sec.	
	Setting range	0 to 10000	
F77	Basis rate for tool life judgment		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. 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. 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. **Note:** The above judgment function is invalid if this parameter is set to 0.
	Program type	M, E	
	Conditions	Immediate	
	Unit	%, min	
	Setting range	0 to 9999	
F78	Selection of separating ratio of graphic display		Selection of separation ratio between side view and front view (or rear view) when two split plane indication mode has been selected. **F78** = 0 ..... 1:1 **F78** = 1 ..... 2:1 **F78** = 2 ..... 5:1 **Example: F78** = 1  NM211-00217
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1, 2	

Classification	USER

Display title	EIA/ISO

Address	Meaning		Description
F79	—		
	Program type	M, E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	Binary, eight digits	
F80	—		
	Program type	M, E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	Binary, eight digits	

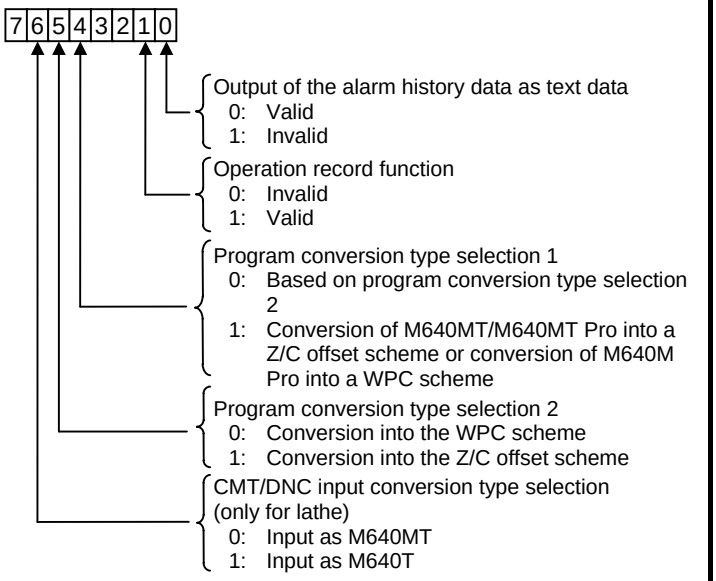
Classification	USER

Display title	EIA/ISO

Address	Meaning		Description
F81	—		7 6 5 4 3 2 1 0 Program management function 0: Normal 1: Editing prohibited (in 9000's) Program management function 0: Normal 1: Dieplaying prohibited (in 9000's) Fixed value (0) Add-in MAZATROL 0: Invalid 1: Valid Program management function 0: Normal 1: Editing prohibited (in 8000, 9000's) Program management function 0: Normal 1: Displaying prohibited (in 8000, 9000's) Retaining the program transfer settings 0: Invalid 1: Valid Input error prevention function 0: Invalid 1: Valid
	Program type	M, E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	Binary, eight digits	
F82	—		7 6 5 4 3 2 1 0 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 0: Invalid 1: Valid Basis for tool life judgment 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 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 0: Invalid 1: Valid **Note:** Bit 1 in parameter **F82** is valid only when an inch/metric unit system change simplifying function (option) is provided.
	Program type	M	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	Binary, eight digits	

Classification	USER

Display title	EIA/ISO

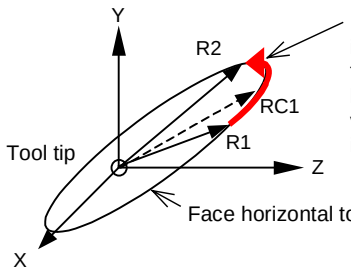
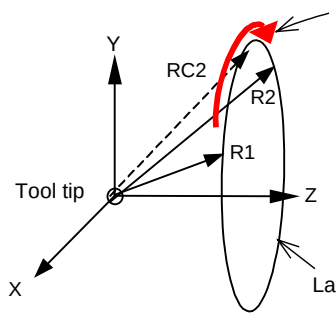
Address	Meaning		Description
F83	—		 **Note:** Bit 6 in **F83** is used to identify the type of program during input from CMT or DNC.
	Program type	M, E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	Binary, eight digits	

Classification	USER

Display title	EIA/ISO

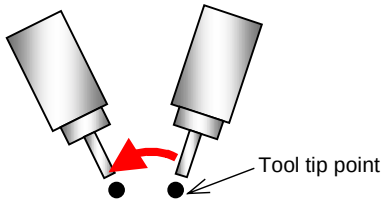
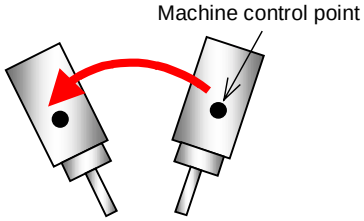
Address	Meaning		Description
F84	—		7 6 5 4 3 2 1 0 Tool offset data is taken into account for the current-position counter during execution of EIA programs 0: No 1: Yes Fixed cycle (B → J) 0: B 1: J Spare tool search for EIA 0: Group number assignment 1: Tool number assignment Timing to validate new 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. Machine coordinate system (G92) selection 0: M32, M PLUS, M640M, or M640M Pro system 1: M2 system 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 Tape operation 0: Not operated until the buffer is full. 1: Operated at a unit of EOB. When no tool data has been designated during EIA/ISO program execution with the MAZATROL tool length data validated. 0: Operation is executed. 1: Alarm state
	Program type	M, E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	Binary, eight digits	

Classification	USER	Display title	EIA/ISO

Address	Meaning	Description
F85	—	7 6 5 4 3 2 1 0 Table rotational machining 0: Always valid 1: Outside the area only Ignorance of radial interference check 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. Tool tip point control scheme 0: Uniaxial rotation interpolation 1: Joint interpolation Fixed value (0) Reset to cancel G68.2 0: Valid 1: Invalid Display of surface definition θ 0: Invalid 1: Valid Output of the B-axis unclamping code before B-axis indexing 0: To output 1: Not to output - bit 3 = 0  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. - bit 3 = 1  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.

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	—	7 6 5 4 3 2 1 0 Output of M250 (Spindle Speed Confirmation) before a turning feed 0: Not to output 1: To output 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 Override scheme for G0 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 Display of the **PART SHAPE** window 0: Display on the X-Y plane 1: Display on the X-Z plane 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 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 Display format of **REMAIN** on the **POSITION** display 0: Displayed in the axial system that actually moves (machine coordinate system). 1: Displayed in the programming coordinate system corresponding to the angle of the B-axis. - bit 2 = 0 Makes the override function valid for the movement of the tool tip point.  - bit 2 = 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	M, E
Conditions	At power on
Unit	Bit
Setting range	Binary, eight digits

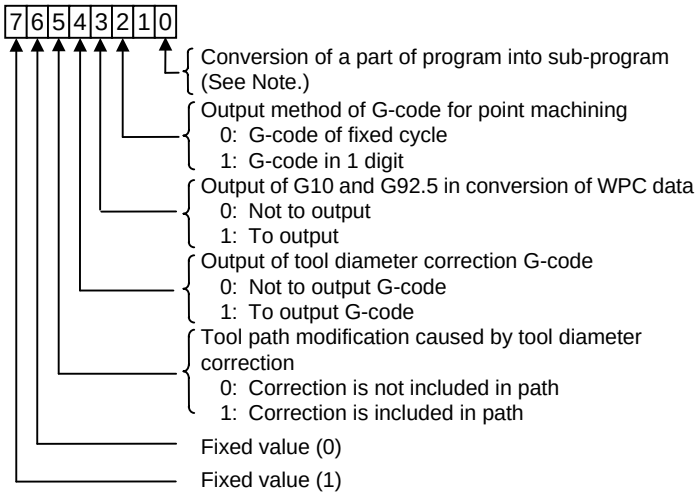
Classification	USER

Display title	EIA/ISO

Address	Meaning		Description
F87	—		7 6 5 4 3 2 1 0 Gap offset type 0: Moves the machine. 1: Does not move the machine. Zero point mismatch check 0: Valid (an alarm is output in case of a mismatch) 1: Invalid Data alteration checking function 0: Invalid 1: Valid
	Program type	M, E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	Binary, eight digits	

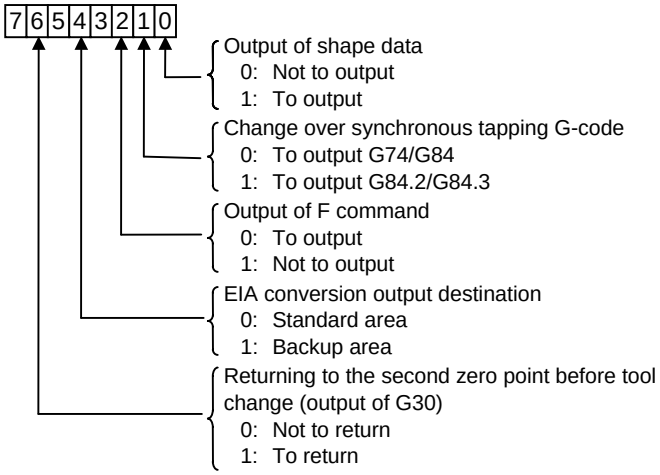
Classification	USER

Display title	EIA/ISO

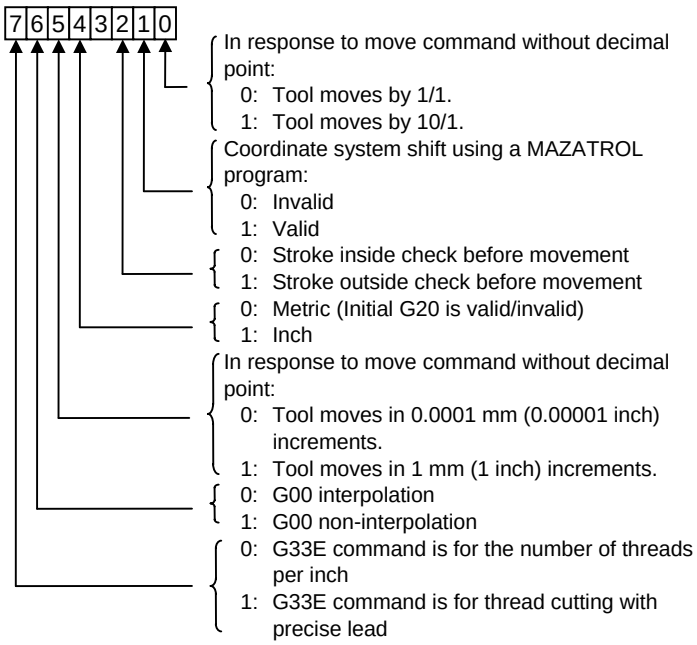
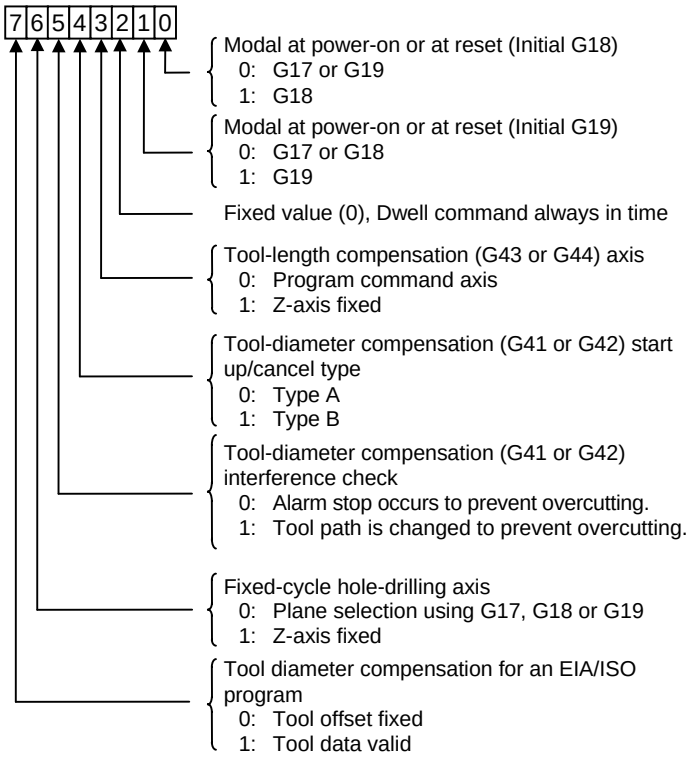
Address	Meaning	Description
F88	—	Set this parameter to specify functions related to the conversion from MAZATROL program into an EIA program.  **Note:** When bit 0 of **F88** is set to 1 on converting to the EIA program, the line machining, groove machining, face machining portions in the MAZATROL program are respectively programmed as subprograms. **Example:** Common unit [1] ⋮ FCE MILL [4] [2] ⋮ LINE CTR [5] [3] ⋮ END **Notes:** **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. **F88** bit 0 { 1: To make subprogram 0: Not to make subprogram Conversion into EIA program Main program (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; % Subprogram N1(UNo.2FACE-MILL); G0X5.Y2.; G1X50.; ⋮ [4] G0X60.Y60.; M99; Unit of machining units N2(UNo.3LINE-CTR); G2X10.Y10.R5.; G1X50.; ⋮ [5] G2X40.Y10.R5.; M99; % Unit of machining units

Program type	M, E
Conditions	Immediate
Unit	Bit
Setting range	Binary, eight digits

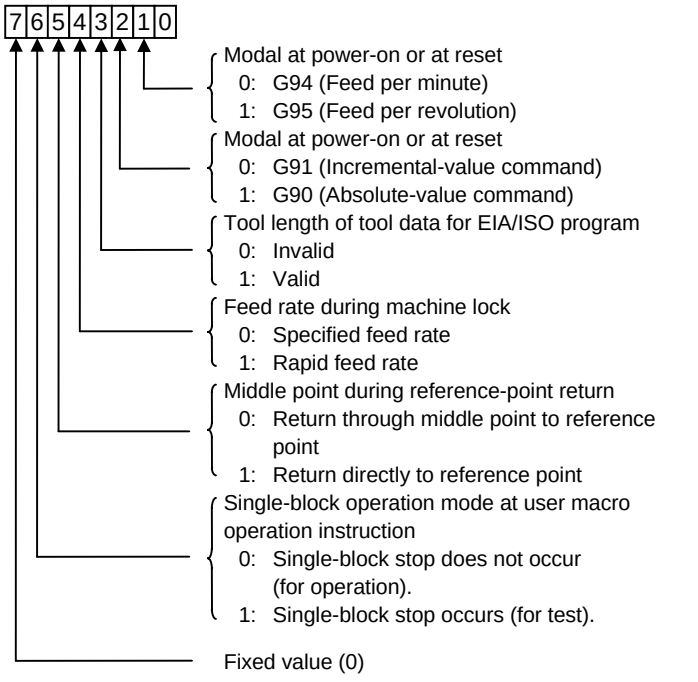
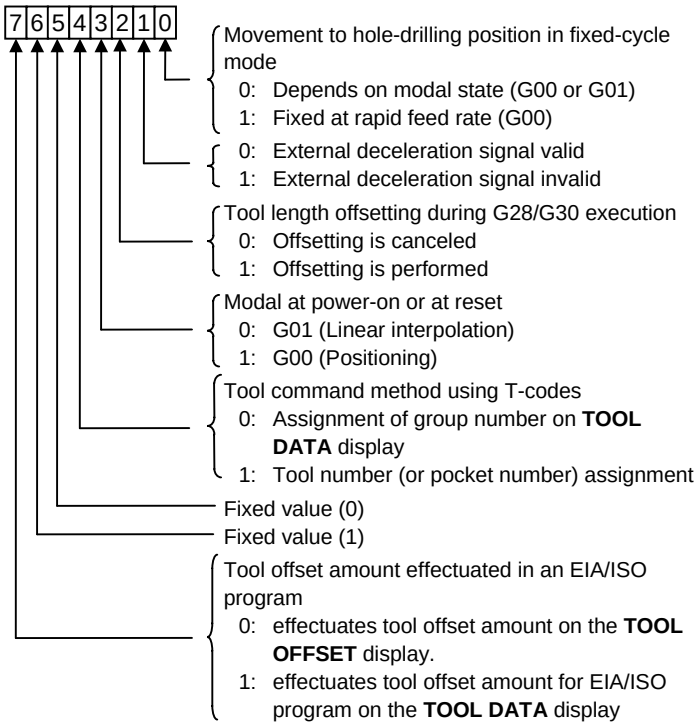
Classification	USER	Display title	EIA/ISO

Address	Meaning		Description
F89	—		Set this parameter to specify functions related to the conversion from MAZATROL program into an EIA program. 
	Program type	M, E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	Binary, eight digits	
F90	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

Classification	USER	Display title	EIA/ISO

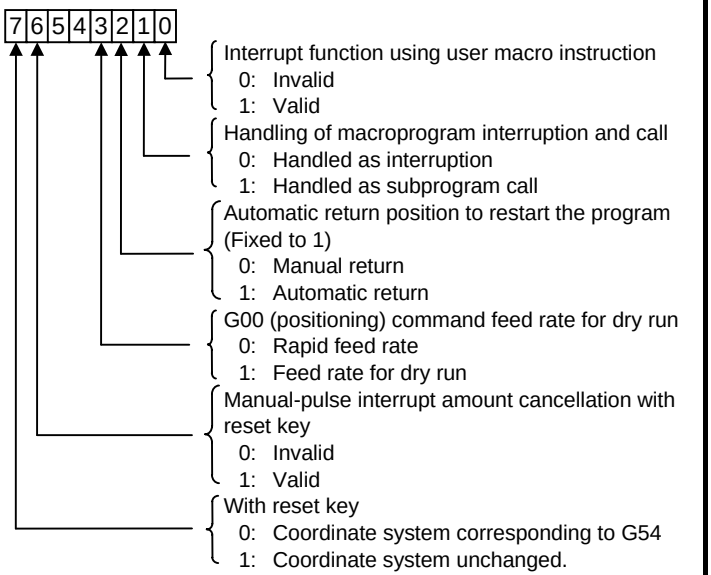
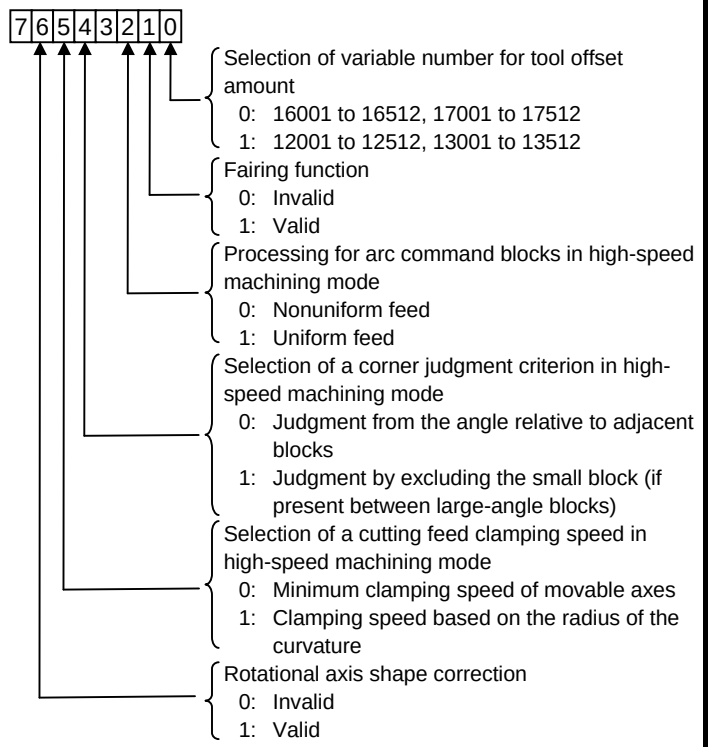
Address	Meaning		Description
F91	—		 **Note:** For changing bit 4 (millimeter/inch system), set **M18** bit 7 to **0** once for zero point return execution if the simplified software OT function is effective (when machine parameter **M18** bit 7 = 1, **R2** bit 7 = 0). (This initialization is required when the simplified OT function is effective even for one axis.)
	Program type	M, E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	Binary, eight digits	
F92	—		
	Program type	M, E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	Binary, eight digits	

Classification	USER	Display title	EIA/ISO

Address	Meaning		Description
F93	—		
	Program type	M, E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	Binary, eight digits	
F94	—		
	Program type	M, E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	Binary, eight digits	

Classification	USER

Display title	EIA/ISO

Address	Meaning		Description
F95	—		
	Program type	M, E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	Binary, eight digits	
F96	—		
	Program type	M, E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	Binary, eight digits	

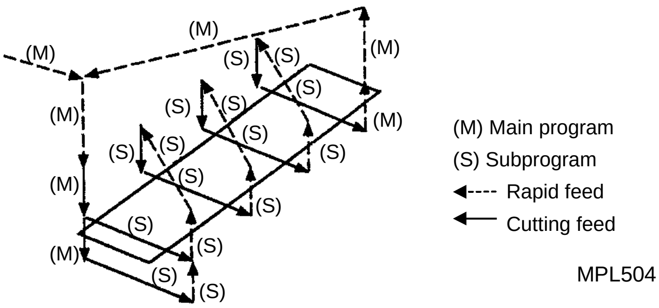
Classification	USER	Display title	EIA/ISO

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.

Setting value	Coordinates system	Setting value	Coordinates system
1	G54	5	G58
2	G55	6	G59
3	G56		
4	G57	Others	G54

F97	
Program type	M, E
Conditions	Immediate
Unit	—
Setting range	0 to 255

Address	Meaning	Description
F98	Number of macro variable to be used in the EIA conversion function	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.



Main program

```
            :
G01Z_;  ←  To the height of cutting face
#_=_;
 |   └─ Height of cutting face
 └───── Macro variable (F98)
M98P_H_;
            :
M30;
%
```

Subprogram

```
N_(_);
     :
G00Z_;   ←  Return on Z-axis
X_Y_;    ←  Moves to the start point of the next machining on Z-axis
G01Z#_;  ←┐
XY_;      │
     :    Machining on Z-axis specified with the variable
M99;
%
```

[Units that use macro variables]

FCE MILL (cutting in one direction), **TOP EMIL**, **POCKET**, **PCKT MT**, **PCKT VLY**

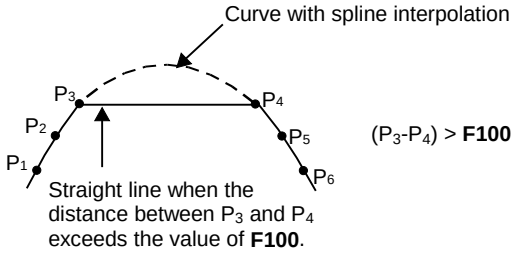
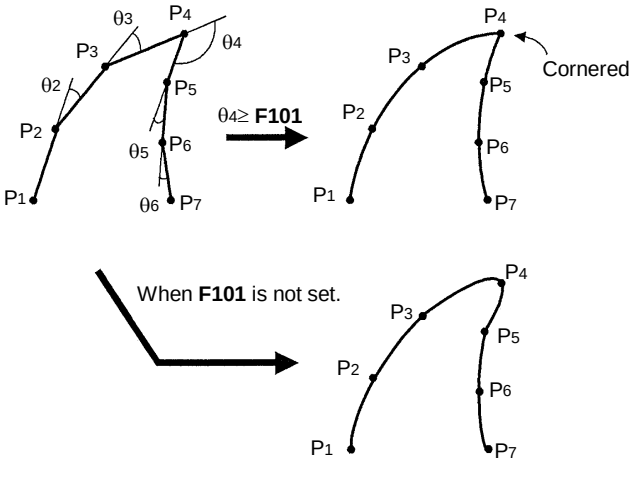
Notes:

1. 3D machining cannot be output using subprograms.

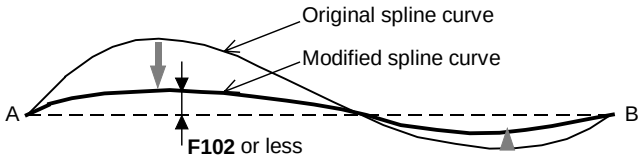
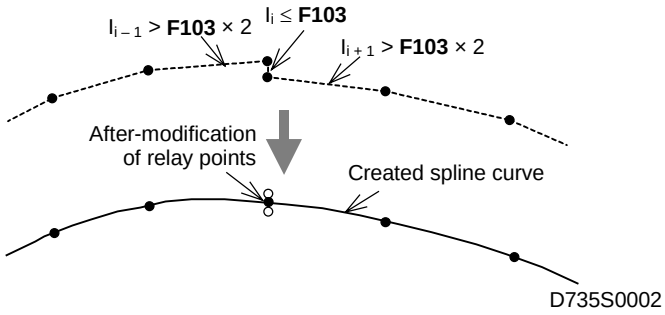
2. Subprogram is output in the absolute mode (G90).

F98	
Program type	M, E
Conditions	Immediate
Unit	—
Setting range	100 to 199 500 to 999

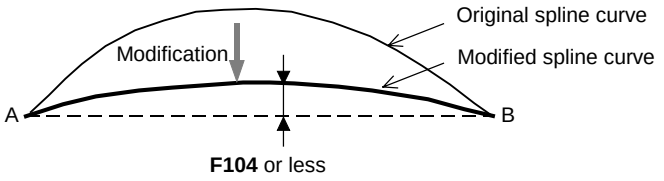
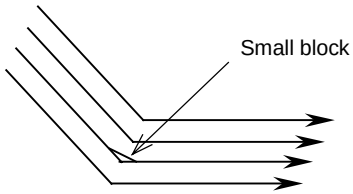
Classification	USER	Display title	EIA/ISO

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.  MPL505
	Program type	E	
	Conditions	Immediate	
	Unit	0.0001 mm/0.00001 inch	
	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.  MPL506
	Program type	E	
	Conditions	Immediate	
	Unit	1°	
	Setting range	0 to 179	

Classification	USER	Display title	EIA/ISO

Address	Meaning		Description
F102	Fine spline interpolation curve error (Block including the point of inflection)		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**.  D735S0001
	Program type	E	
	Conditions	Immediate	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to 99999999	
F103	Spline interpolation fairing block length		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. Suppose that the i-th block in the fine spline interpolation mode has a block length of l_i: If l_{i-1} is greater than **F103** × 2 l_i is equal to or less than **F103** l_{i+1} is greater than **F103** × 2 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.  D735S0002 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. Refer to the relevant specification for further details. This parameter is effective when bit 1 of **F96** is 1.
	Program type	E	
	Conditions	Immediate	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to 99999999	

Classification	USER	Display title	EIA/ISO

Address	Meaning		Description
F104	Fine spline interpolation curve error (Block including no inflection point)		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**.  D735S0003
	Program type	E	
	Conditions	Immediate	
	Unit	0.0001mm/0.00001 inch	
	Setting range	0 to 99999999	
F105 **F106**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
F107	Small block judgment length		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. Judgment length for judging the small block  This parameter is valid when bit 4 of **F96** is 1.
	Program type	E	
	Conditions	Immediate	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to 99999999	
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	1°	
	Setting range	0 to 99999999	

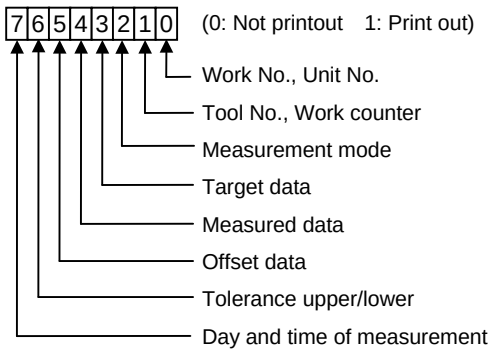
Classification	USER	Display title	EIA/ISO

Address	Meaning		Description
F109 **F110**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
F111 (bit 0)	Selection of display type of tapping tool in solid mode		In the solid mode of the **TOOL PATH CHECK** 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	
F111 (bit 1)	Use/disuse of dry run during thread cutting		**F111** (bit 1) = 0: Disuse of dry run during thread cutting cycle **F111** (bit 1) = 1: Use of dry run during thread cutting cycle
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
F111 (bit 2)	Use/disuse of feed hold during thread cutting		**F111** (bit 2) = 0: Disuse of feed hold during thread cutting cycle **F111** (bit 2) = 1: Use of feed hold during thread cutting cycle
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	

Classification	USER	Display title	EIA/ISO

Address	Meaning		Description
F111 (bit 3)	Direction of rotation of the C-axis during C-axial threading with G01.1		Select the direction of rotation of the C-axis during C-axial threading based on G01.1. **F111** (bit 3) = 0: The C-axis rotates CW (forward). **F111** (bit 3) = 1: The C-axis rotates CCW (backward).
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
F111 (bit 4)	EIA tool command suffix valid/invalid		Specify whether the assignment of the tool identification code (suffix) by the T-command is valid or invalid. **F111** (bit 4) = 0: The EIA tool command suffix is invalid. **F111** (bit 4) = 1: The EIA tool command suffix is valid.
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
F111 (bit 5)	Tool correction amount selection for EIA/ISO programs		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. **F111** (bit 5) = 0: Wear correction data is not added **F111** (bit 5) = 1: Wear correction data is added
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
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	

Classification	USER	Display title	EIA/ISO

Address	Meaning		Description
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 integratingly.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
F113 (bit 1)	Data handling on the milling tool of a group that has expired in tool life		**F113** (bit 1) = 0: Operation will be continued. **F113** (bit 1) = 1: Operation will be stopped.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	

Classification	USER

Display title	EIA/ISO

Address	Meaning		Description
F113 (bit 2)	Data handling on the turning tool of a group that has expired in tool life		**F113** (bit 2) = 0: Operation will be continued. **F113** (bit 2) = 1: Operation will be stopped.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
F113 (bit 3)	Tool life management of the FLASH tool		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.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
F113 (bit 4)	Tool life management – Life time		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.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
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	

Classification	USER	Display title	EIA/ISO

Address	Meaning		Description
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} (rpm). **F114** (bit 0) = 1: The maximum selectable C-axial cutting feed rate is 400 min^{-1} (rpm).
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
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)		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. .
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	

Classification	USER	Display title	EIA/ISO

Address	Meaning		Description
F114 (bit 2)	—		Tool shape check during tool measurement **F114** (bit 2) = 0: During measurement for IN and EDGE tools, alarm **653 ILLEGAL TOOL DESIGNATED** occurs since the measurements are possible only at 0 degrees of the B-axis. **F114** (bit 2) = 1: The shape of the tool is not checked.
	Program type	M, E	
	Conditions	At power on	
	Unit	—	
	Setting range	0, 1	
F114 (bit 3)	—		Moving axes by using G49 (tool length cancel) in G43 (tool length offset) mode **F114** (bit 3) = 0: Valid **F114** (bit 3) = 1: Invalid
	Program type	E	
	Conditions	At power on	
	Unit	—	
	Setting range	0, 1	
F114 (bit 4)	—		Selecting a rethreading function **F114** (bit 4) = 0: Using the rethreading function **F114** (bit 4) = 1: Not using the rethreading function
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
F114 (bit 5)	—		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	

Classification	USER	Display title	EIA/ISO

Address	Meaning		Description
F114 (bit 6)	—		**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)	—		**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 inch/min	
	Setting range	0 to 65535	
F116	Feed rate of the threading runout – X-axis		Specify the X-axial feed rate for the runout of the threading cycle.
	Program type	M, E	
	Conditions	Immediate	
	Unit	1 mm/min	
	Setting range	0 to 240000	

Classification	USER	Display title	EIA/ISO

Address	Meaning		Description
F117	Feed rate of the threading runout – Y-axis		Specify the Y-axial feed rate for the runout of the threading cycle.
	Program type	M, E	
	Conditions	Immediate	
	Unit	1 mm/min	
	Setting range	0 to 240000	
F118	Feed rate of the threading runout – Z-axis		Specify the Z-axial feed rate for the runout of the threading cycle.
	Program type	M, E	
	Conditions	Immediate	
	Unit	1 mm/min	
	Setting range	0 to 240000	
F119	Runout feed rate for the inside diameter threading cycle		Specify the runout feed rate for the inside diameter threading cycle. This parameter is valid only when chamfering is not specified in the inside diameter threading unit of the MAZATROL program.
	Program type	M	
	Conditions	Immediate	
	Unit	1 mm/min	
	Setting range	0 to 240000	
F120	Clamping speed for the threading cycle – X-axis		Specify the X-axial clamping speed for the threading cycle.
	Program type	M, E	
	Conditions	Immediate	
	Unit	1 mm/min	
	Setting range	0 to 99999999	

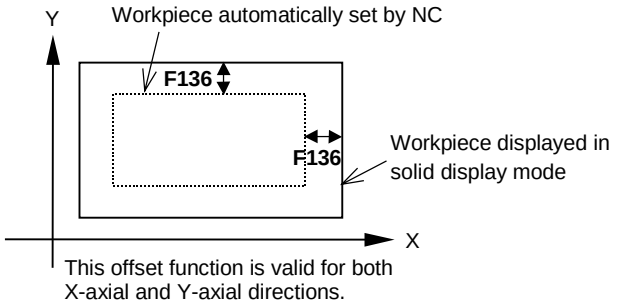
Classification	USER	Display title	EIA/ISO

Address	Meaning		Description
F121	Clamping speed for the threading cycle – Y-axis		Specify the Y-axial clamping speed for the threading cycle.
	Program type	M, E	
	Conditions	Immediate	
	Unit	1 mm/min	
	Setting range	0 to 99999999	
F122	Clamping speed for the threading cycle – Z-axis		Specify the Z-axial clamping speed for the threading cycle.
	Program type	M, E	
	Conditions	Immediate	
	Unit	1 mm/min	
	Setting range	0 to 99999999	
F123	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
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 OFFFSET**: GEOMETRIC OFFSET
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	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		The data input in the following items are checked based on the permissible data alteration amount set in this parameter. **TOOL DATA**: WEAR COMP. X, WEAR COMP. Y, WEAR COMP. Z **TOOL OFFFSET**: WEAR COMP.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 99999999	
F126 to **F132**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
F133	Pitch of tapping tool for display in detail in solid mode		Set the pitch of tapping tool displayed when the tapping tool is displayed in detail (**F111** bit 0 = 1) in solid mode on **TOOL PATH CHECK** display.
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 mm/0.01 inch	
	Setting range	0 to 65535	
F134	Thread depth of tapping tool for display in detail in solid mode		Set the thread depth of tapping tool displayed when the tapping tool is displayed in detail (**F111** bit 0 = 1) in solid mode on **TOOL PATH CHECK** display.
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 mm/0.01 inch	
	Setting range	0 to 65535	

Classification	USER	Display title	EIA/ISO

Address	Meaning		Description
F135	Tool-drawing accuracy in solid mode		Specify tool-drawing accuracy in the solid mode of the **TOOL PATH CHECK** display. As the specified value is greater (maximum value: 9), accuracy increases progressively.
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 9	
F136	Amount of offset for dummy workpiece shape in solid mode		It is possible to change the size of the workpiece created automatically when solid mode is selected on the **TOOL PATH CHECK** display. 
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 mm/0.01 inch	
	Setting range	–99999999 to 99999999	
F137	Number of jaws displayed in solid mode for No. 1 turning spindle		Specify for the solid mode of the **TOOL PATH CHECK** display the number of jaws displayed automatically. If 0 is entered, three jaws will be displayed as standard.
	Program type	M	
	Conditions	Immediate	
	Unit	Jaws	
	Setting range	0 to 9	
F138	Number of jaws displayed in solid mode for No. 2 turning spindle		Specify for the solid mode of the **TOOL PATH CHECK** display the number of jaws displayed automatically. If 0 is entered, three jaws will be displayed as standard.
	Program type	M	
	Conditions	Immediate	
	Unit	Jaws	
	Setting range	0 to 9	

Classification	USER	Display title	EIA/ISO

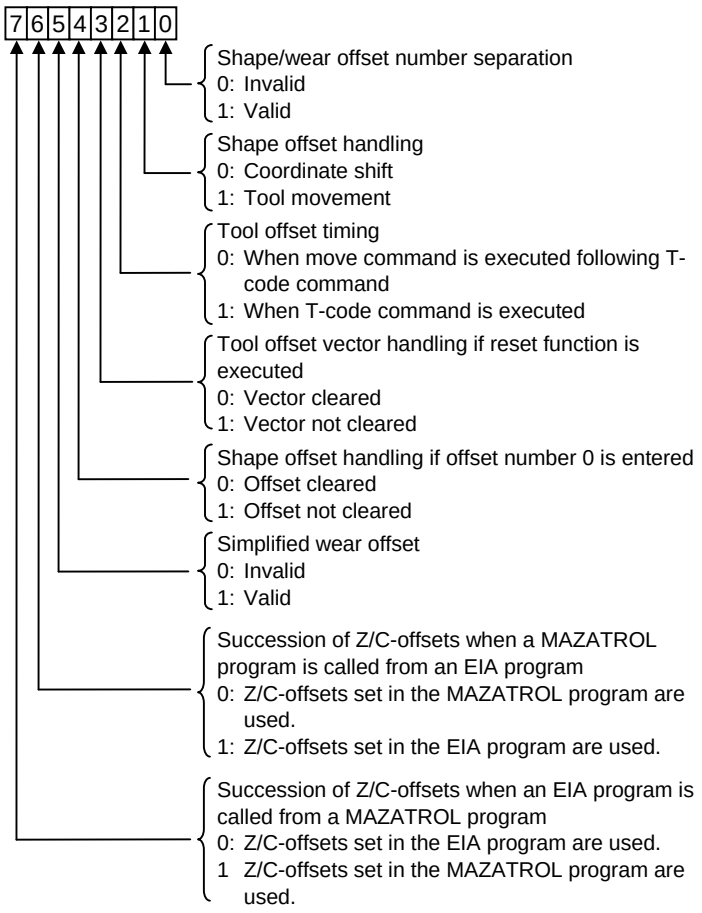
Address	Meaning		Description
F139	Angle offset for the jaws displayed in solid mode for the No. 1 turning spindle		Specify for the solid mode of the **TOOL PATH CHECK** display the amount of C-axial angle offset for the jaws displayed automatically.
	Program type	M	
	Conditions	Immediate	
	Unit	0.1°	
	Setting range	0 to 3599	
F140	Angle offset for the jaws displayed in solid mode for the No. 2 turning spindle		Specify for the solid mode of the **TOOL PATH CHECK** display the amount of C-axial angle offset for the jaws displayed automatically.
	Program type	M	
	Conditions	Immediate	
	Unit	0.1°	
	Setting range	0 to 3599	
F141 to **F144**	—		Not used
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
F145	Rapid feed override when data alteration is detected		The rapid feed rate override is reduced to the value set here, when data alteration is detected. This value will be handled as100 if 0 is set here.
	Program type	M	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 100	

Classification	USER

Display title	EIA/ISO

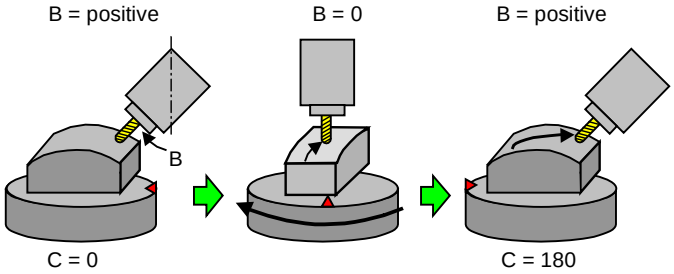
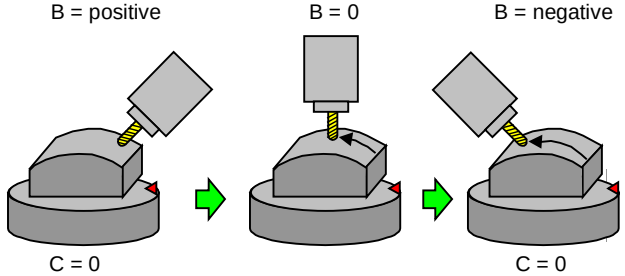
Address	Meaning		Description
F146 to **F153**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
F154	—		Parameter for system internal setting Setting prohibited
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
F155 to **F160**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

Classification	USER	Display title	EIA/ISO

Address	Meaning	Description
F161	—	Lathing tool offset functions  **F161** bit 0: T○○□□ command: - For offset number separation ○○ 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.

Program type	M
Conditions	Immediate
Unit	Bit
Setting range	Binary, eight digits

Classification	USER	Display title	EIA/ISO

Address	Meaning	Description
F162	—	7 6 5 4 3 2 1 0 During independent start of tool tip point control 0: Movement according to the particular amount of offset 1: No movement according to the particular amount of offset Type of passage of tool tip point through singular point Chamfer/corner R-command address selection 0: (I), (K), (R), (,R), (,C), (,A) 1: (R), (C), (A) Fixed hole-machining cycle return selection 0: Initial point 1: R-point 6 digits in T-command for turning Use of the M Pro scheme as the method of selecting the Length correction axis bit 0: Invalid 1: Valid MAZATROL program check for missing Z-offset 0: Valid 1: Invalid Encoder polarity selection 0: Pulse rate increase for forward rotation command 1: Pulse rate reduction for forward rotation command **F162** bit 1 - Type of passage of tool tip point through singular point 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. **Example:**  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.) **Example:** 

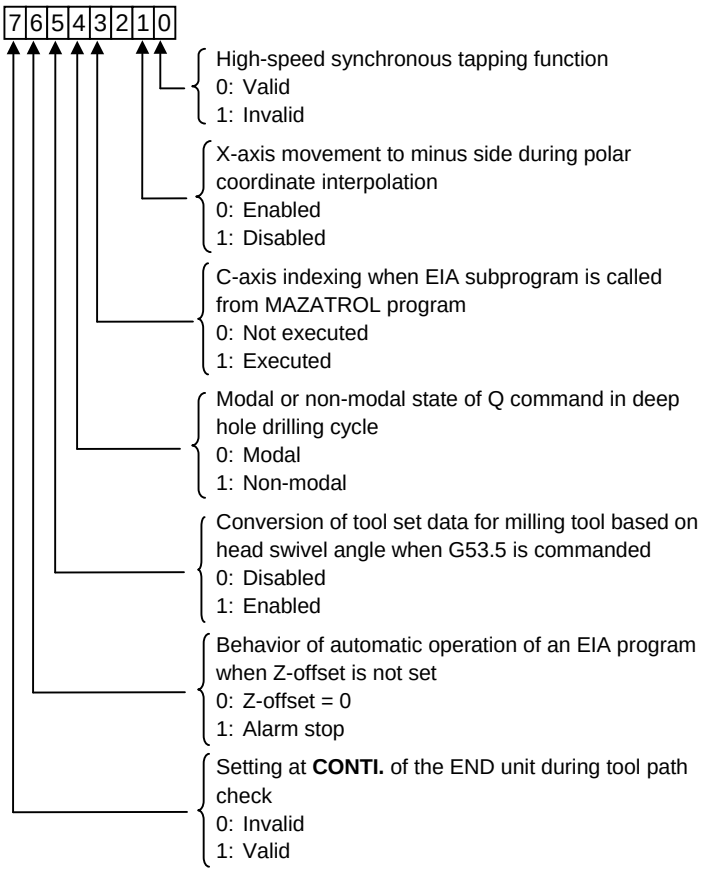
Program type	M, E
Conditions	After stop of movement
Unit	Bit
Setting range	Binary, eight digits

Classification	USER

Display title	EIA/ISO

Address	Meaning		Description
F163	—		7 6 5 4 3 2 1 0 Bar feeder scheduling function 0: Invalid 1: Valid Incorporation of wear offset data into the current position display in EIA/ISO program mode 0: Invalid 1: Valid Incorporation of wear offset data into the current position display in MAZATROL program mode 0: Valid 1: Invalid Position of thread turning tool nose on solid drawing 0: Tool tip 1: Position measured using TOOL EYE Barrier check on solid drawing 0: Invalid 1: Valid Menu on the **DATA I/O** display (floppy disk) 0: Hidden 1: Displayed Menu on the **DATA I/O** display (tape) 0: Hidden 1: Displayed Menu on the **DATA I/O** display (CMT) 0: Hidden 1: Displayed
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	Binary, eight digits	
F164	—		Automatic tool data setting conditions 7 6 5 4 3 2 1 0 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 **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
	Program type	M	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	Binary, eight digits	

Classification	USER	Display title	EIA/ISO

Address	Meaning	Description
F165	—	 Supplemental explanation of **F165** Bit 4 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. 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 **ILLEGAL NUMBER INPUT** occurs. 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.

Program type	M
Conditions	After stop of movement
Unit	Bit
Setting range	Binary, eight digits

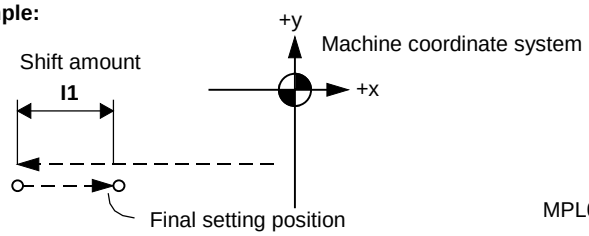
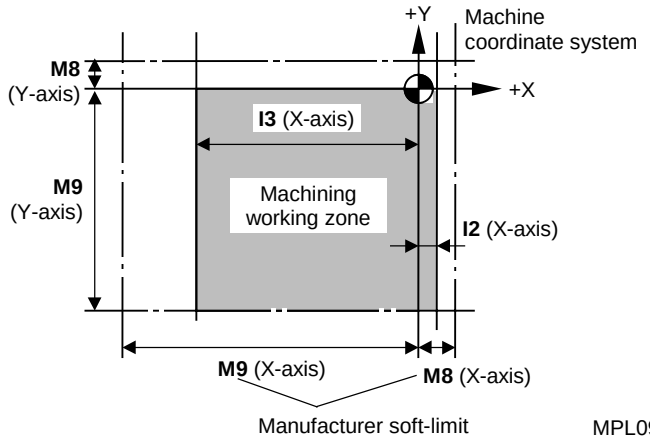
Classification	USER

Display title	EIA/ISO

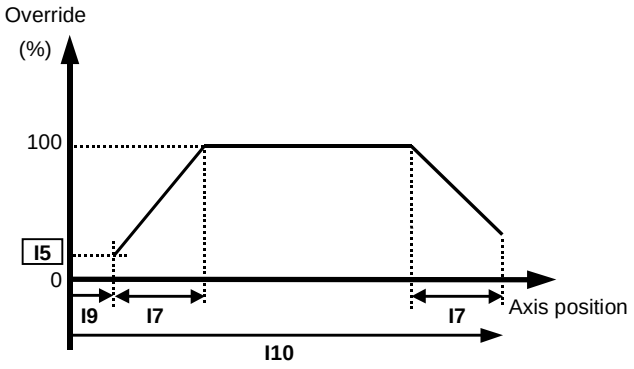
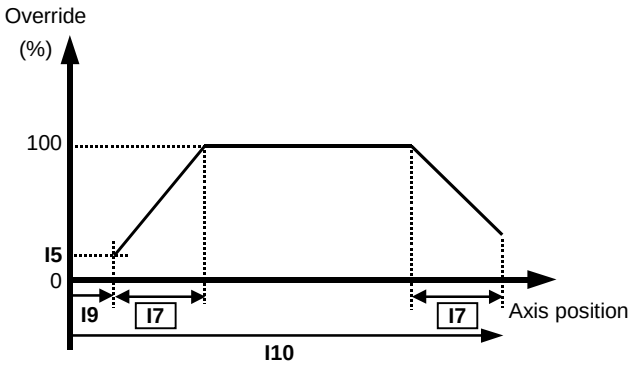
Address	Meaning		Description
F166	—		7 6 5 4 3 2 1 0 Alteration of tool set value (tool length) on the **TOOL DATA** display in the automatic operation mode 0 : Enabled 1 : Disabled Type of wear offset indicated in the milling tool list on the **TOOL DATA** display 0 : Cutting edge offset 1 : Wear offset ID No./Tool name selection on the **TOOL DATA** display 0 : ID No. 1 : Tool Name
	Program type	M	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	Binary, eight digits	
F167 **F168**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

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

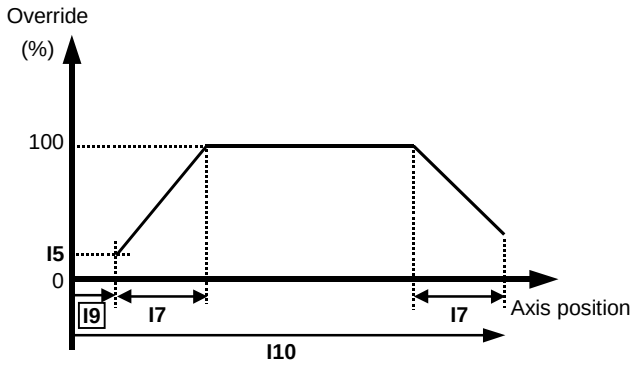
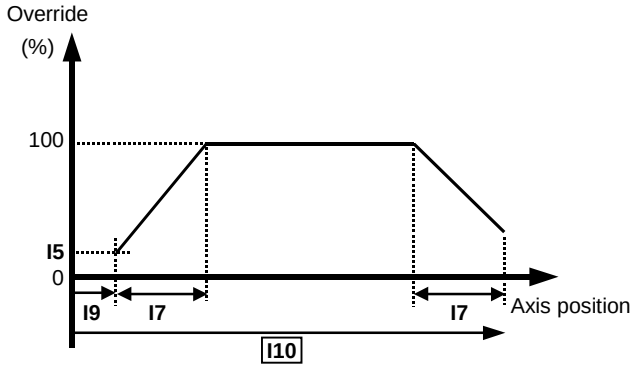
Classification	USER	Display title	SOFT LIMIT

Address	Meaning		Description
I1	Shift amount of unidirectional positioning (G60)		The amount and direction of shift from the final setting position during unidirectional positioning of the point-machining or during execution of G60. **I1** < 0: Positioning in minus direction **I1** > 0: Positioning in plus direction **Example:**  MPL091 **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.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	0.0001 mm/0.00001 inch (0.0001°)	
	Setting range	0 to ±99999999	
I2	Upper (plus direction) user soft-limit		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. **Example:**  MPL092 If the machine is likely to overstep its working zone, an alarm will occur and the machine will stop. **Notes:** **1.** These parameters are valid only when bit 2 of **I14** is 0. **2.** These parameters are invalid if **I2** = **I3**. **3.** 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 inch (0.0001°)	
	Setting range	0 to ±99999999	
I3	Lower (minus direction) user soft-limit		
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	0.0001 mm/0.00001 inch (0.0001°)	
	Setting range	0 to ±99999999	

Classification	USER	Display title	SOFT LIMIT

Address	Meaning		Description
I4	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
I5	Function for making the G0 speed variable Variable override: Minimum value		
	Program type	M, E	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 100	
I6	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
I7	Function for making the G0 speed variable Variable control area		
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.0001 mm/0.00001 inch/ 0.0001°	
	Setting range	0 to 99999999	

Classification	USER	Display title	SOFT LIMIT

Address	Meaning		Description
I8	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
I9	Function for making the G0 speed variable Variable control area lower limit		
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.0001 mm/0.00001 inch/ 0.0001°	
	Setting range	0 to 99999999	
I10	Function for making the G0 speed variable Variable control area upper limit		
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.0001 mm/0.00001 inch/ 0.0001°	
	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.
	Program type	E	
	Conditions	After stop of movement	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to ±99999999	

Classification	USER	Display title	SOFT LIMIT

Address	Meaning		Description
I12	Clamping value for the amount of handle interruption		Set the clamping value for the amount of handle interruption.
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm 0.0001 inch 0.0001 deg	
	Setting range	0 to 99999999	
I13	—		7 6 5 4 3 2 1 0 Execution of G28 (reference-point return): 0: Memory-type zero-point return 1: Watchdog-type zero-point return 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 Removal of control axes 0: No (Not removed) 1: Yes (Removed)
	Program type	M, E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	Binary, eight digits	
I14	—		7 6 5 4 3 2 1 0 Mirror image with respect to the machine zero-point 0: Invalid 1: Valid User software limits (I2, I3) 0: Valid 1: Invalid Tool-tip relief after spindle orientation during execution of G75, G76, G86 or point-machining (boring or back-boring) 0: Required 1: Not required Direction of the relief mentioned above 0: Plus 1: Minus
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	Binary, eight digits	

Classification	USER

Display title	SOFT LIMIT

Address	Meaning		Description
I15 to **I24**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

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	

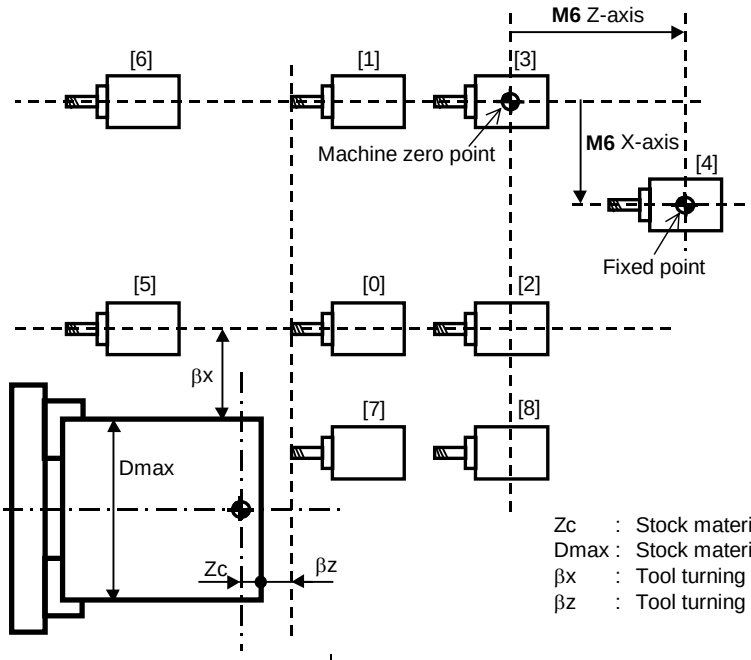
Classification	USER

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 position specification code	Specify tool change position from [0] through [8] below.

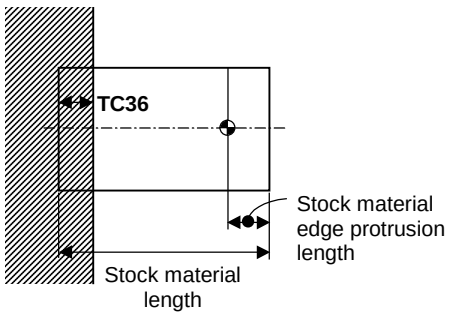


Zc : Stock material edge projection length
Dmax : Stock material maximum outside diameter
βx : Tool turning clearance (X-axis) **TC35**
βz : Tool turning clearance (Z-axis) **TC36**

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

Note:
SU10 = 5 or 6, Z-axis tool change 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 **TC36**.



Program type	M
Conditions	Immediate
Unit	—
Setting range	0 to 8

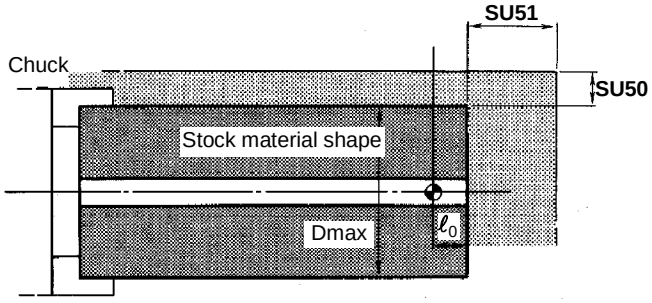
Classification	USER	Display title	SYSTEM

Address	Meaning		Description
SU11	Movement of axes during approach		Specify the types of axes to be simultaneously moved when approaching the tool to the next machining area. 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. <Supplementary description> 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.
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
SU12	Rotating position specified in the index unit after tool change		Select the rotating position specified in the index unit after tool change. 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.
	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		Set the display color for the tool nose marks in each system indicated on the **TOOL PATH CHECK** and **TRACE** displays. 0 : Red (default) 1 : Green 2 : Blue 3 : Yellow 4 : Pink 5 : Cyan 6 : White
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	—	
	Setting range	0 to 6	

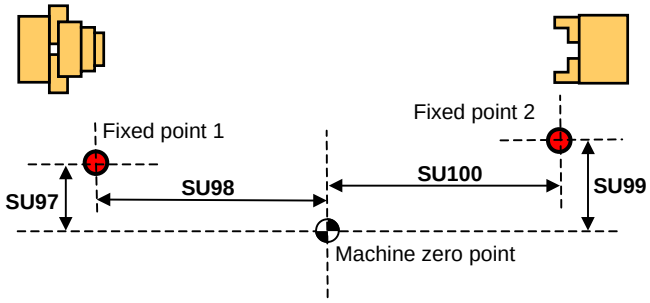
Classification	USER	Display title	SYSTEM

Address	Meaning		Description
SU15	Name of thrust axis for W-axis		Specify the name of the axis used as the thrust axis for the W-axis.
	Program type	E	
	Conditions	After stop of movement	
	Unit	—	
	Setting range	&41 to &5A	
SU16	Movement to C-axis index swivel position when Z-offset scheme is used		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.
	Program type	M	
	Conditions		
	Unit	—	
	Setting range	0, 1	
SU17 to **SU48**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
SU49	Delay timer for the parts catcher		Set dwell time for the parts catcher.
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 sec	
	Setting range	0 to 9999	

Classification	USER	Display title	SYSTEM

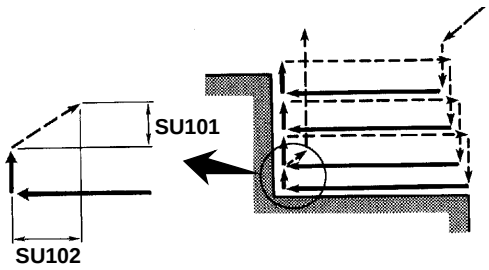
Address	Meaning		Description
SU50	Tool turning clearance (radial value) in X-axis		Tool turning clearance is required to prevent interference between the tool and stock material during tool change in automatic operation.  NM211-00220 Dmax: Stock material maximum outside diameter ℓ_0: Stock material edge projection length
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 65535	
SU51	Tool turning clearance in Z-axis		
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 65535	
SU52 **SU53**	Lower-turret retraction function Tool number of the retraction tool		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
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 960	
SU54 to **SU96**	—		
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

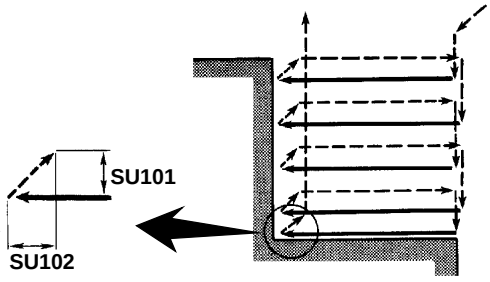
Classification	USER	Display title	SYSTEM

<table>
<tr><th>Address</th><th colspan="2">Meaning</th><th>Description</th></tr>
<tr><td rowspan="5">SU97
to
SU100</td><td colspan="2" rowspan="5">Lower-turret retraction function
Fixed point of the retraction position</td><td rowspan="5">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.
 

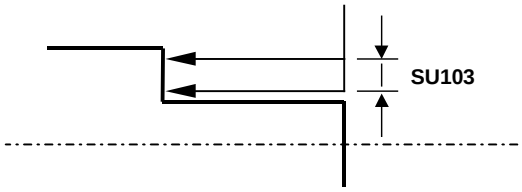
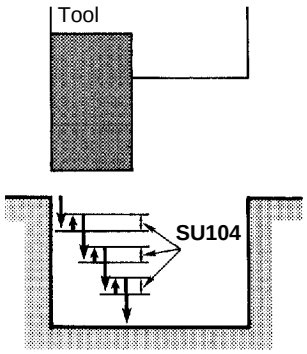
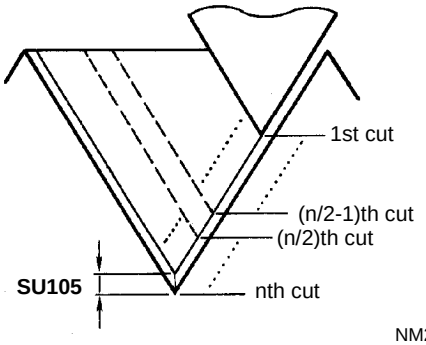
SU97: X-axis retraction point – Fixed point 1
SU98: Z-axis retraction point – Fixed point 1
SU99: X-axis retraction point – Fixed point 2
SU100: Z-axis retraction point – Fixed point 2

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.

Note:
Set this parameter for the system of the lower turret.</td></tr>
<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 inch</td></tr>
<tr><td>Setting range</td><td>0 to 99999999</td></tr>
<tr><td rowspan="5">SU101</td><td colspan="2" rowspan="5">Return distance (radial value) in X-axis at wall during rough cutting in bar machining or in corner machining of EIA/ISO program</td><td rowspan="9">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.
 

Example 2:
High speed rough cutting
Escape will be made by SU101 and SU102 specified distance during return after reaching the wall.
 
</td></tr>
<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 inch</td></tr>
<tr><td>Setting range</td><td>0 to 65535</td></tr>
<tr><td rowspan="5">SU102</td><td colspan="2" rowspan="5">Return distance (radial value) in Z-axis at wall during rough cutting in bar machining or in corner machining of EIA/ISO program</td></tr>
<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 inch</td></tr>
<tr><td>Setting range</td><td>0 to 65535</td></tr>
</table>

Classification	USER	Display title	SYSTEM

Address	Meaning		Description
SU103	Cutting depth in the composite-type fixed cycle (G271/G272, G71/G72)		If a cutting depth has not been specified in the program, operation will occur in accordance with the setting of this parameter. 
	Program type	E	
	Conditions	Immediate	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to 65535	
SU104	Pecking return distance in groove cutting unit and grooving (G274/G275, G74/G75)		 NM211-00248
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.0001 mm/rev 0.00001 inch/rev	
	Setting range	0 to 65535	
SU105	Cut depth (diametral value) for final cut in thread cutting unit Cut depth (diametral value) for final cut in composite-type thread cutting cycle G276, G76		 NM211-00247
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to 65535	
SU106	Minimum cut depth clamping value in thread cutting unit and composite-type thread cutting cycle G276		Clamping will follow the setting of **SU106** if the calculated value of the cut depth with the threading unit is smaller than the setting of **SU106**. This parameter is valid only for the infeed operation of the fixed-area scheme. 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.
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to 65535	

Classification	USER	Display title	SYSTEM

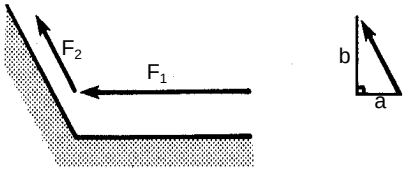
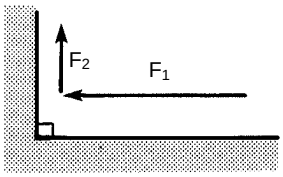
Address	Meaning		Description
SU107	Safety clamp speed		Set the spindle safety clamp speed of a tool whose diameter (nominal diameter) is not specified in the tool data. This clamp speed is invalid for the tools with tool diameter (or nominal diameter, if diameter cannot be specified) setting in the tool 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. Relational expression for maximum allowable spindle speed N and tool diameter D $N = \frac{60 \times 10^3}{\pi D} \sqrt{\frac{2E}{m}}$ (rpm) Safeguarding strength: E (J) Tool diameter: D (mm) Tip mass: m (kg)
	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	
SU110 to **SU152**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

Classification	USER	Display title	SYSTEM

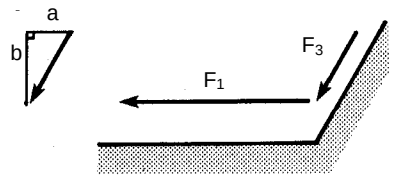
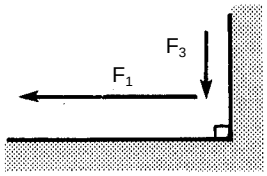
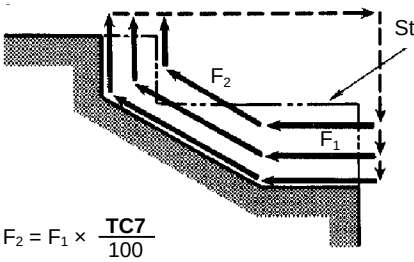
<table>
<tr><th>Address</th><th colspan="2">Meaning</th><th>Description</th></tr>
<tr><td rowspan="5">SU153
(bit 0)</td><td colspan="2">—</td><td>- If the command block of G284/G288 (machining G-codes, F30 = 0) or of G84/G88 (turning G-codes, F30 = 1) does not contain spindle forward/reverse rotation M-codes, one of the following M-codes will be output, depending on the status of bit 0 in parameter SU153:
<table>
<tr><td colspan="2"></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>
- 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.</td></tr>
<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>—</td></tr>
<tr><td>Setting range</td><td>0, 1</td></tr>
<tr><td rowspan="5">SU154
to
SU168</td><td colspan="2">—</td><td rowspan="5">Invalid</td></tr>
<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>

2-3-7 User parameter TURNING (TC)

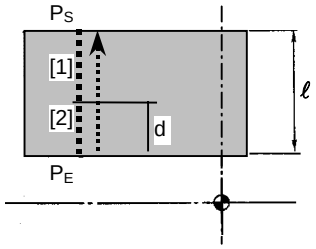
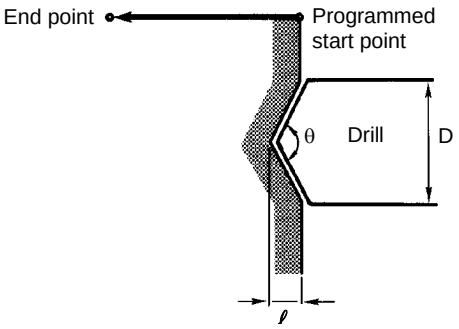
Classification	USER	Display title	TURNING

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 $A = T \times \frac{\mathbf{TC1}}{100}$ 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		 NM211-00262 $F_2 = F_1 \times \frac{\mathbf{TC2}}{100} \times \left\lvert \frac{b}{a} \right\rvert$ F_1 : Feed rate for rough cutting F_2 : Increased feed rate This is effective only when **TC141** (bit 0) = 1.
	Program type	M	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 500	
TC3	Acceleration rate in up-going wall slope (90°) for rough cutting in bar machining unit		F_1 : Feed rate for rough cutting F_2 : Increased feed rate  NM211-00263 $F_2 = F_1 \times \frac{\mathbf{TC3}}{100}$ **Note:** Up to 500 % can be set. This is effective only when **TC141** (bit 0) = 1.
	Program type	M	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 500	
TC4	Selection of escape pattern from wall (90°) in rough cutting cycle		This parameter will be used to select escape pattern (0, 1 or 2) when wall is vertical in G71/G72 mode. **TC4** = 0: Identical with ordinary path **TC4** = 1: Escape at 45° from wall **TC4** = 2: Feed rate accelerated at wall Accelerated feed rate F is expressed as follows. $F = F_0 \times \frac{\mathbf{TC3}}{100}$ (where F_0 = Feed rate specified in program) (⇨**TC3**)
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 2	

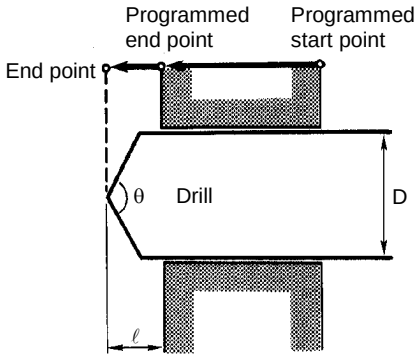
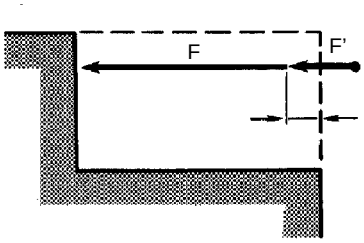
Classification	USER	Display title	TURNING

Address	Meaning		Description
TC5	Deceleration rate in down-going taper for rough cutting in bar machining unit		 NM211-00264 $F_3 = F_1 \times \frac{\mathbf{TC5}}{100} \times \left\lvert \frac{a}{b} \right\rvert$ F_1 : Feed rate for rough cutting F_3 : Reduced feed rate This is effective only when **TC141** (bit 0) = 1.
	Program type	M	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 500	
TC6	Deceleration rate in down-going wall slope (90°) for rough cutting in bar machining unit		 F_1 : Feed rate for rough cutting F_3 : Reduced feed rate NM211-00265 $F_3 = F_1 \times \frac{\mathbf{TC6}}{100}$ This is effective only when **TC141** (bit 1) = 1.
	Program type	M	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 500	
TC7	Acceleration rate on outside stock contour for rough cutting in copy machining unit		 F_1 : Feed rate inside stock contour F_2 : Feed rate outside stock contour $F_2 = F_1 \times \frac{\mathbf{TC7}}{100}$ NM211-00266
	Program type	M	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 500	
TC8	Acceleration pitch error ratio in thread cutting unit		Used to calculate acceleration distance in thread cutting unit $L = L_0 \left\{ -\ell_n \left(\frac{\mathbf{TC8}}{1000}\right) - 1 + \frac{\mathbf{TC8}}{1000} \right\}$ L : Acceleration distance L_0 : Distance over which feed rate become constant
	Program type	M	
	Conditions	Immediate	
	Unit	0.1%	
	Setting range	0 to 100	

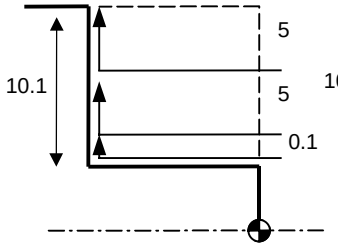
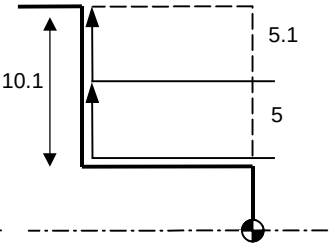
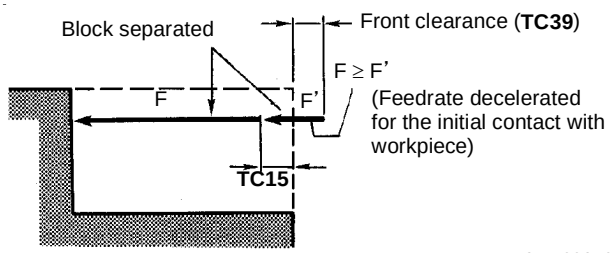
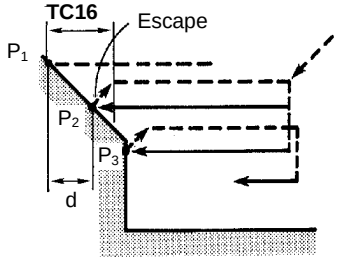
Classification	USER	Display title	TURNING

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

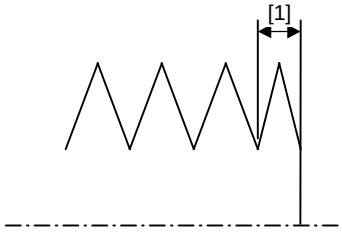
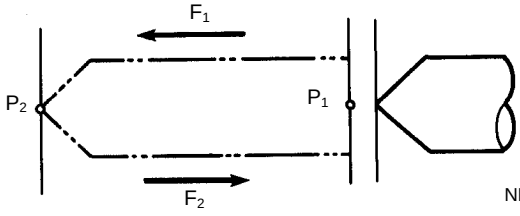
Classification	USER	Display title	TURNING

Address	Meaning		Description
TC12	Deceleration rate at cutting end in turning-drilling unit		$F' = F \times \frac{\textbf{TC12}}{100}$ F : Specified feed rate F' : Feed rate at cutting start  Feed rate at F' to the point distant by ℓ from the end point NM211-00269 $\begin{cases} \ell = \frac{D}{2 \times \tan\theta/2} + \frac{D}{10} & (0° < \theta < 80°) \\ \ell = \frac{D}{10} & (\theta \geq 180°) \end{cases}$
	Program type	M	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 100	
TC13	Deceleration rate at rough cutting start in bar machining unit and copy machining unit		 Deceleration clearance (**TC15**) NM211-00270 $F' = F \times \frac{\textbf{TC13}}{100}$ F: Specified feed rate F': Feed rate at cutting start (⇨**TC15**)
	Program type	M	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 100	

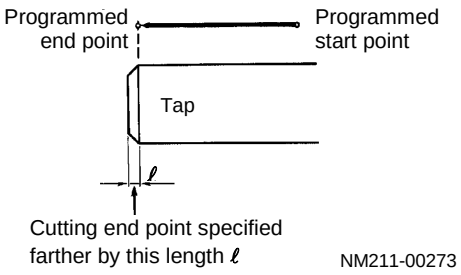
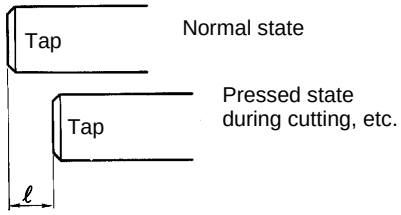
Classification	USER	Display title	TURNING

Address	Meaning		Description
TC14	Maximum permissible rate of increase of the initial cutting depth during roughing		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. **Example 1:** **TC14** = 0 (R-depth.: 5)  Cutting to a depth of 10.1 is executed in 3 rounds of cutting (5, 5, and 0.1). **Example 2:** **TC14** = 10% (R-depth.: 5)  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.
	Program type	M	**Note:** Up to 100% can be set. When a value larger than 100% is set, it is regarded as 0.
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 100	
TC15	Deceleration clearance at start of rough cutting in bar machining unit and copy machining unit		 NM211-00256
	Program type	M	**Note:** Using this parameter will reduce load in the initial contact between the tool and workpiece. (⇨**TC13**)
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 65535	
TC16	Tolerance for escape in high speed rough cutting cycle of bar machining unit		 d : Distance in cutting direction between P_1 and P_2 NM211-00257
	Program type	M	If **TC16** ≥ d, tool escape from the workpiece. If **TC16** < d, tool doesn't escape from the workpiece.
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 65535	

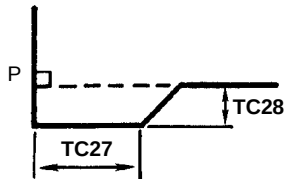
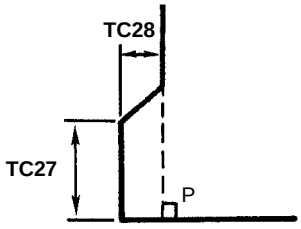
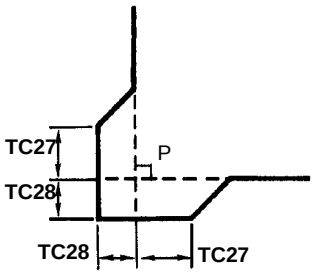
Classification	USER	Display title	TURNING

Address	Meaning		Description
TC17	Pitch error correction during threading acceleration		Set the starting pitch error rate of threading.  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 $d_1 = D \times \frac{\mathbf{TC19}}{100}$ 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		 NM211-00271 $F_2 = F_1 \times \frac{\mathbf{TC20}}{100}$ 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	

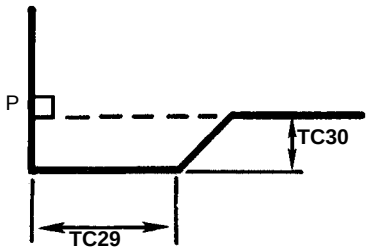
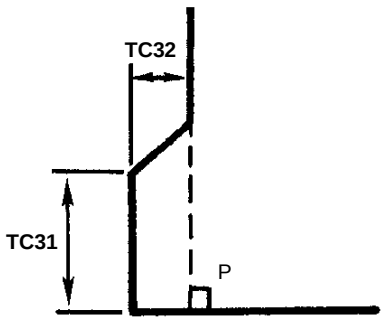
Classification	USER	Display title	TURNING

Address	Meaning		Description
TC21	Incomplete threading portion length calculation coefficient for turning-tap tip		$\ell = P \times \frac{TC21}{10}$ P : Tapping pitch ℓ : Incomplete thread portion length  NM211-00273
	Program type	M	
	Conditions	Immediate	
	Unit	Pitch/10	
	Setting range	0 to 99	
TC22	Turning-tapper elongation calculation coefficient		$\ell = P \times \frac{T\ C22}{10}$ P : Tapping pitch ℓ : Tapper elongation  NM211-00274
	Program type	M	
	Conditions	Immediate	
	Unit	Pitch/10	
	Setting range	0 to 99	
TC23	Thread height calculation coefficient for outside diameter, face/rear thread cutting (metric)		$h = P \times \frac{TC23}{10000}$ 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)		$h = P \times \frac{TC24}{10000}$ h : Thread height P : Thread pitch
	Program type	M	
	Conditions	Immediate	
	Unit	0.01%	
	Setting range	0 to 65535	

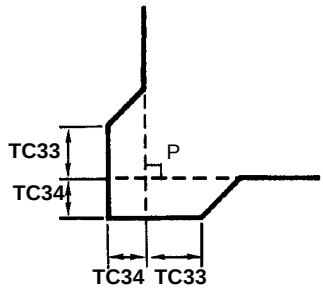
Classification	USER	Display title	TURNING

Address	Meaning		Description
TC25	Thread height calculation coefficient for outside diameter, face/rear thread cutting (inch)		$h = P \times \frac{\mathbf{TC25}}{10000}$ h : Thread height P : Thread pitch
	Program type	M	
	Conditions	Immediate	
	Unit	0.01%	
	Setting range	0 to 65535	
TC26	Thread height calculation coefficient for inside diameter thread cutting (inch)		$h = P \times \frac{\mathbf{TC26}}{10000}$ h : Thread height P : Thread pitch
	Program type	M	
	Conditions	Immediate	
	Unit	0.01%	
	Setting range	0 to 65535	
TC27	Recessing width for #1 to #3		<#1>  NM211-00277 <#2>  NM211-00278
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 65535	
TC28	Recessing depth #1 to #3		<#3>  NM211-00279 P: Programmed end point
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 65535	

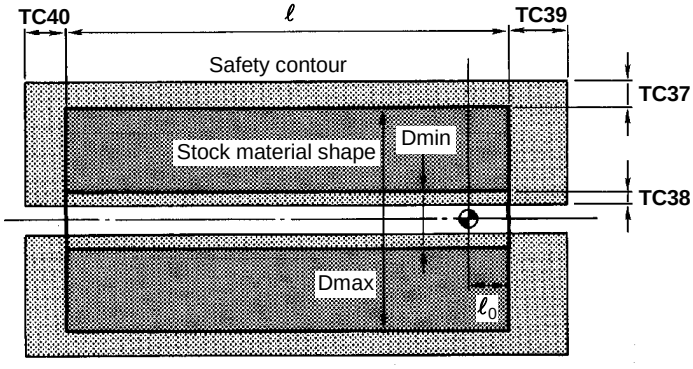
Classification	USER	Display title	TURNING

Address	Meaning		Description
TC29	Recessing width for #4		<#4>  NM211-00280 P: Programmed end point
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 65535	
TC30	Recessing depth for #4		
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 65535	
TC31	Recessing width for #5		<#5>  NM211-00278 P: Programmed end point
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 65535	
TC32	Recessing depth for #5		
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 65535	

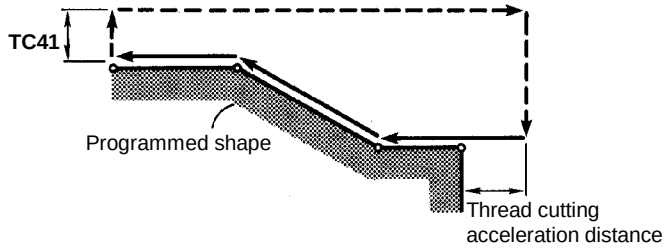
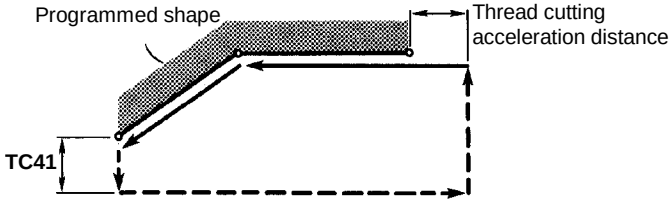
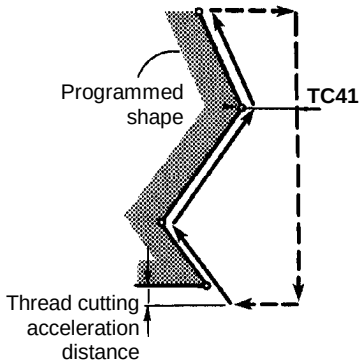
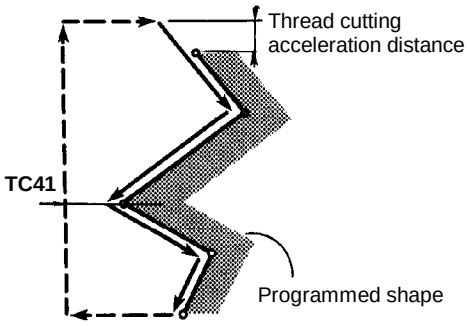
Classification	USER	Display title	TURNING

Address	Meaning		Description
TC33	Recessing width for #6		<#6>  NM211-00279 P: Programmed end point
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 65535	
TC34	Recessing depth for #6		
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 65535	
TC35 **TC36**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

Classification	USER	Display title	TURNING

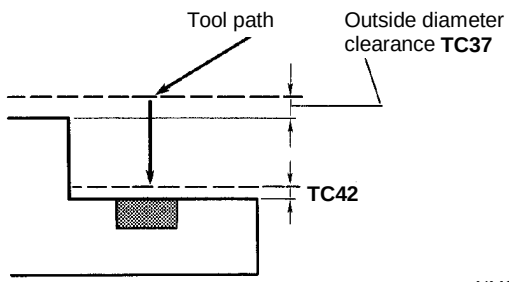
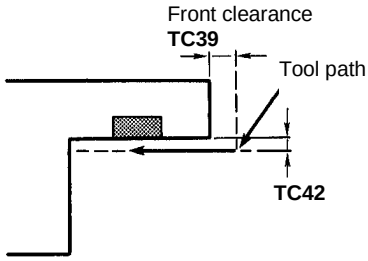
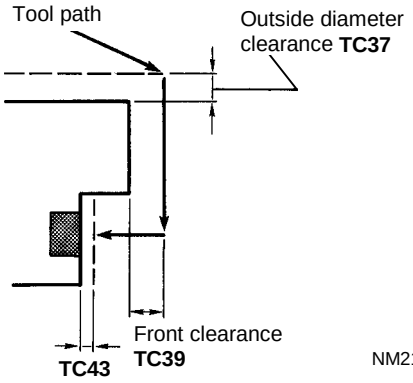
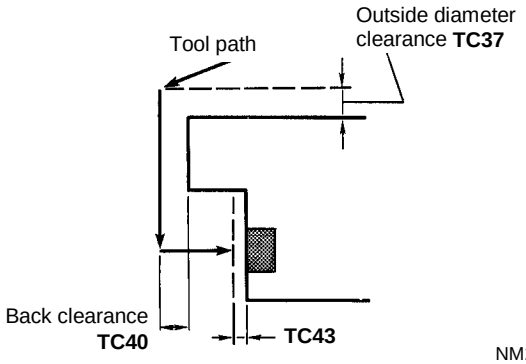
Address	Meaning		Description
TC37	Safety contour clearance — Outside diameter clearance (radial value)		Safety contour clearance is provided for outside of the stock material shape specified by common data in program. Tool approach and escape paths for each unit will be automatically determined according to set data (outside diameter, inside diameter, front clearance, back clearance) for parameters from **TC37** to **TC40**.  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
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	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 inch	
	Setting range	0 to 65535	
TC39	Safety contour clearance — Front clearance		
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 65535	
TC40	Safety contour clearance — Back clearance		
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 65535	

Classification	USER	Display title	TURNING

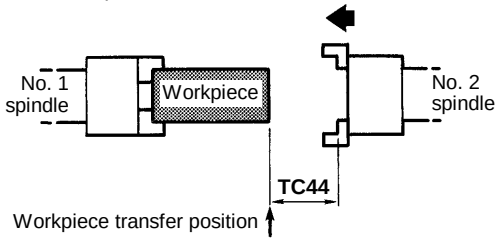
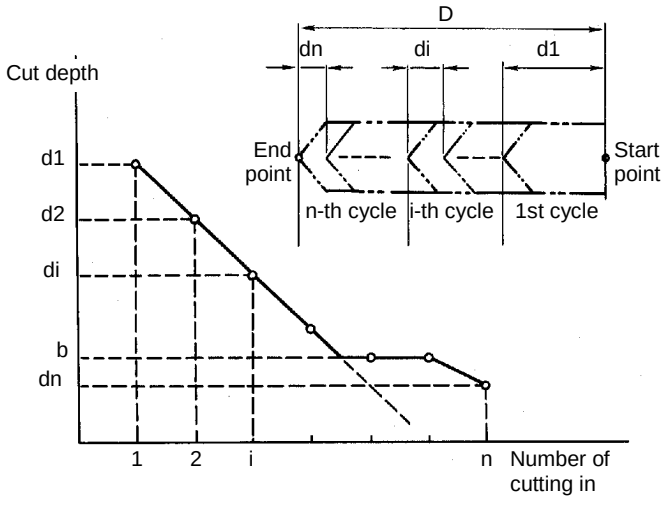
Address	Meaning	Description
TC41	Thread cutting clearance (radial value)	Thread cutting clearance is provided to specify tool return distance for each cycle in thread cutting unit. Thread cutting clearance will be added to the highest portion of thread and repeating path will be determined automatically. <**OUT**>  NM211-00222 <**IN**>  NM211-00223 <**FCE**>  NM211-00224 <**BAK**>  NM211-00225

Program type	M
Conditions	Immediate
Unit	0.001 mm/0.0001 inch
Setting range	0 to 65535

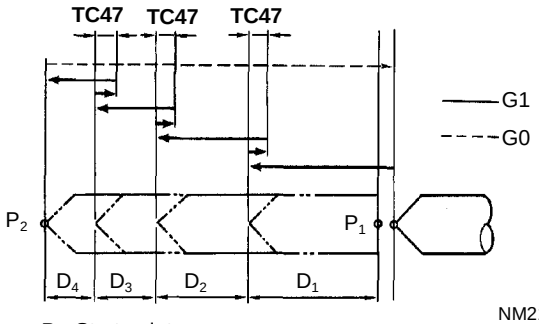
Classification	USER	Display title	TURNING

Address	Meaning		Description
TC42	Groove cutting clearance (radial value) in X-axis		Groove cutting clearance is provided at machining start portion in groove cutting unit. <**OUT**>  NM211-00226 <**IN**>  NM211-00227
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 65535	
TC43	Groove cutting clearance in Z-axis		<**FCE**>  NM211-00228 <**BAK**>  NM211-00229
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 65535	

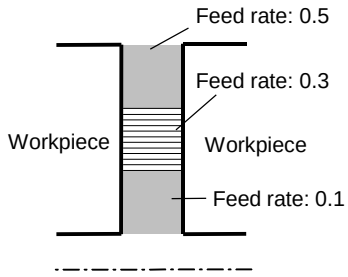
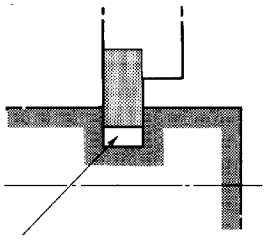
Classification	USER	Display title	TURNING

Address	Meaning		Description
TC44	Workpiece transfer clearance		Workpiece transfer clearance is provided to specify workpiece transfer position in workpiece transfer unit.  NM211-00231 No. 2 spindle traverses by rapid feed rate to a position distant from transfer position by clearance distance, and then transfer operation is initiated. (⇨**TC57**, **TC58**, **TC59**)
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 65535	
TC45	Amount of edge clearance after roughing in the edge-machining unit		After edge-machining unit roughing, this parameter works instead of safety contour clearance FCE parameter **TC39**. If, however, **TC45** is zero, then **TC39** is used.
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 65535	
TC46	Drilling depth decrement in turning-drilling unit		 NM211-00251 D : Drilling depth d1 : Cut depth in 1st cycle di : Cut depth in i-th cycle d1 – **TC46** × (i – 1) (di ≥ b) b (di < b) b : Drilling depth clamping value (**TC48**) (⇨**TC48**)
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 65535	

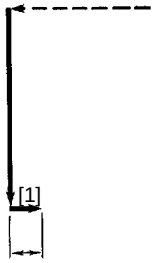
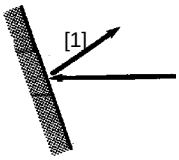
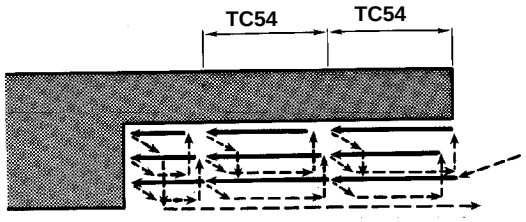
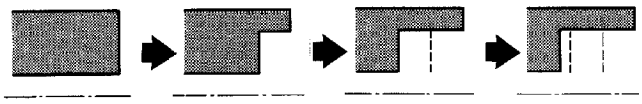
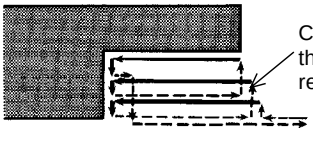
Classification	USER	Display title	TURNING

Address	Meaning		Description
TC47	Pecking return distance in turning-drilling unit		 NM211-00252 P_1: Start point P_2: End point D_n (n = 1 to 4) = Cut depth
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 65535	
TC48	Drilling cut depth clamp value in turning-drilling unit		Minimum turning-drilling cut depth is set. (⇨**TC46**)
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	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} (rpm)	
	Setting range	0 to 65535	

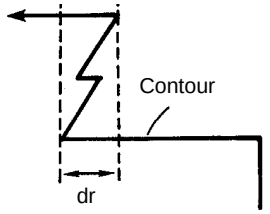
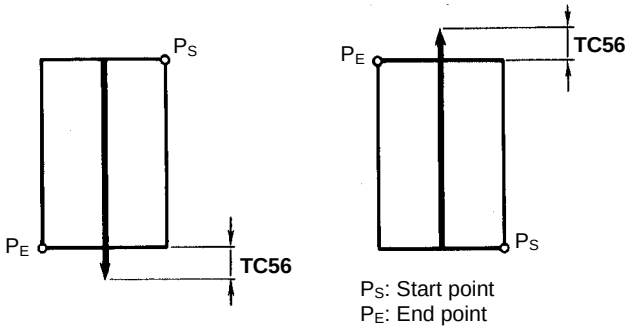
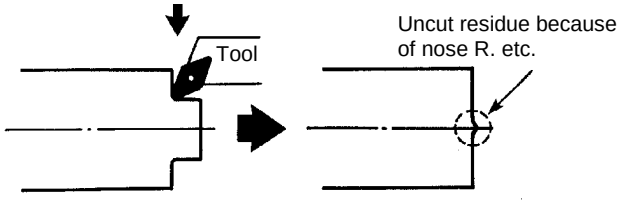
Classification	USER	Display title	TURNING

Address	Meaning		Description
TC50	Number of times that the feed rate is to be reduced during the #4 and #5 cutting-off cycles of a grooving unit		The staring 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. **Example:** Feed rate set at feed item in unit data = 0.5 Feed rate set at roughness item in sequence data = 0.1 **TC50** = 3  When 0 or 1 is set for **TC50**, the feed rate is unchanged.
	Program type	M	
	Conditions	Immediate	
	Unit	Times	
	Setting range	0 to 65535	
TC51	Dwell at the hole bottom during non-through hole drilling cycle of the turning-drilling unit		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.
	Program type	M	
	Conditions	Immediate	
	Unit	Revolutions	
	Setting range	0 to 255	
TC52	Dwell (specification of spindle rotation number) at groove bottom in groove cutting unit		Tool will stop at groove bottom while spindle rotates N times when **TC52** is set to N (N=0 to 255).  Remaining at groove bottom until the spindle rotates N times. NM211-00218
	Program type	M	
	Conditions	Immediate	
	Unit	Revolutions	
	Setting range	0 to 255	

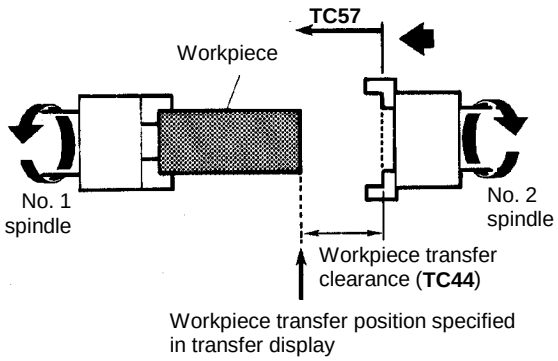
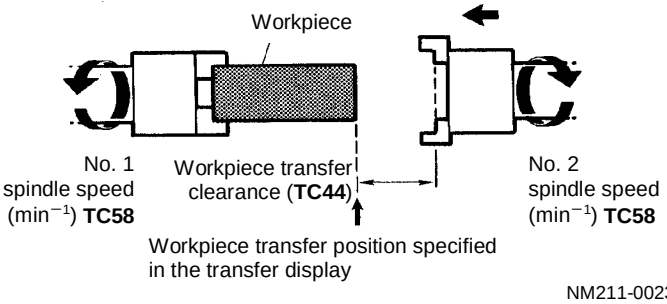
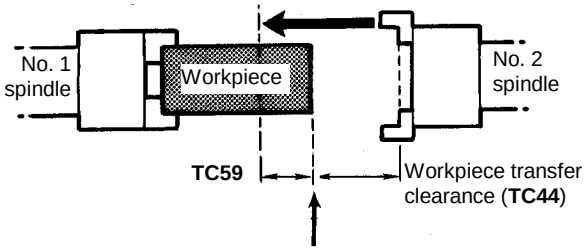
Classification	USER	Display title	TURNING

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 distance Escape in rough cutting of bar machining unit  [1]: Feed speed in this block specified by G01 (**TC53**) NM211-00238
	Program type	M	
	Conditions	Immediate	
	Unit	1 mm/min / 0.1 inch/min	(⇨**TC67**, **TC68**)
	Setting range	0 to 65535	
TC54	Cut depth per cycle for machining inside diameter in bar machining unit		Inside diameter enlarging cycle  NM211-00241 Cutting is promoted gradually from the edge, and machining chip removal efficient.  cf. Standard inside diameter cutting  Cutting to specified depth once through, and machining chip removal not efficient NM211-00242
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 65535	

Classification	USER	Display title	TURNING

Address	Meaning		Description
TC55	Reverse feed tolerance for contour machining		**Example:** Outside diameter machining in normal (– Z-axis) direction  NM211-00243 dr ≤ **TC55** ..... No alarm dr > **TC55** ..... Alarm
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 65535	
TC56	Overtravelling in X-axis direction in edge machining unit		 P_S: Start point P_E: End point NM211-00244 **Note:** By setting an adequate value for **TC56**, uncut residue will not be produced in edge machining.  NM211-00245
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 65535	

Classification	USER	Display title	TURNING

Address	Meaning		Description
TC57	Workpiece pressing speed in workpiece transfer unit		**Example:** Workpiece transferred from No. 1 spindle to No. 2 spindle  NM211-00236 Set this parameter equal to or as close as possible to its standard value. Excessive setting may cause a contact error. (⇨**TC44, TC59**)
	Program type	M	
	Conditions	Immediate	
	Unit	1 mm/min 0.1 inch/min	
	Setting range	0 to 65535	
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  NM211-00237
	Program type	M	
	Conditions	Immediate	
	Unit	min^{-1} (rpm)	
	Setting range	0 to 65535	
TC59	Workpiece pressing distance in workpiece transfer unit		Workpiece transferred from No. 1 spindle to No. 2 spindle  NM211-00255
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/min 0.0001 inch/min	
	Setting range	0 to 65535	

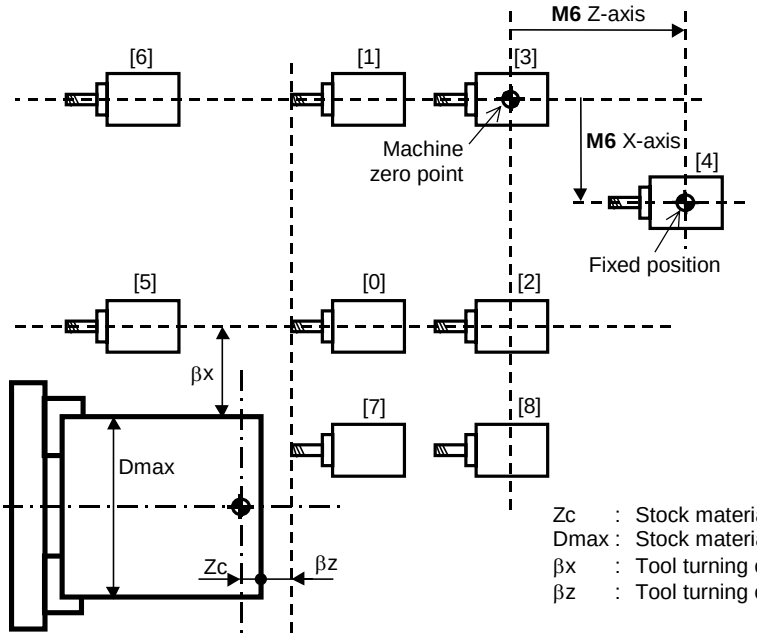
Classification	USER

Display title	TURNING

Address	Meaning		Description
TC60	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
TC61	Simultaneous operation pattern for transfer		Simultaneous operation pattern for transfer of workpieces between two unit jobsites **TC61** = 1: Rotation of the spindle and movement of the Z-axis **TC61** = 2: Orientation of the spindle and movement of the Z-axis **TC61** = 4: Positioning of the C-axis and movement of the Z-axis **Note:** To combine patterns, set the sum total of setting numbers corresponding the conditions.
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 7	

Classification	USER	Display title	TURNING

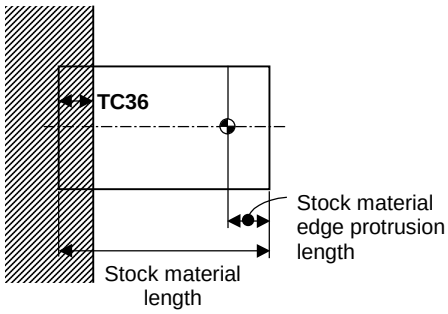
Address	Meaning	Description
TC62	Selection of tool change position specification code for FLASH tool	When a FLASH tool is used, specify the position for indexing the tool, namely, the tool change position for indexing the tool of the same TNo. at the same B-axis angle during programmed operation. In all other cases, the tool change position specified by **SU10** becomes valid.



Zc : Stock material edge projection length
Dmax : Stock material maximum outside diameter
βx : Tool turning clearance (X-axis) **TC35**
βz : Tool turning clearance (Z-axis) **TC36**

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

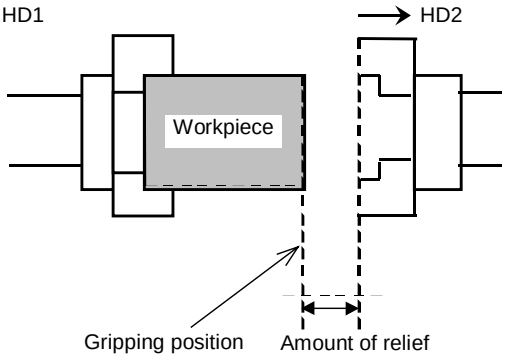
Note:
When **TC62** = 5 or 6, Z-axis tool change position is identical with the end point of previous machining. In the case below, however, this may not be applied.
As shown here, if the longest tool comes into the hatched portion, the position will escape in Z-axis direction by the distance determined by **TC36**.



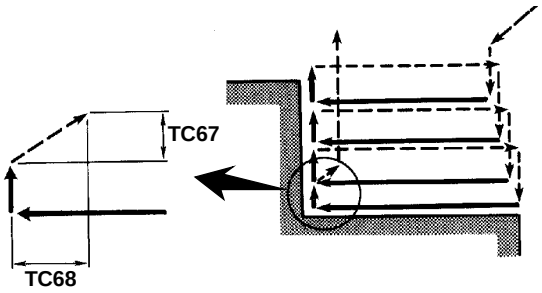
Program type	M
Conditions	Immediate
Unit	—
Setting range	0 to 8

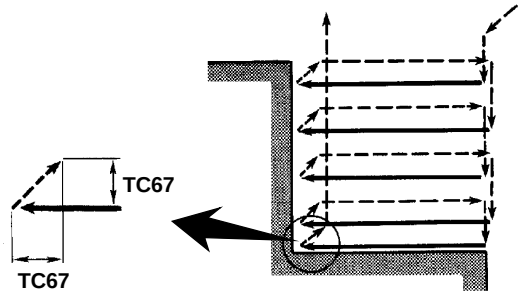
(⇨**TC35, TC36, M5**)

Classification	USER	Display title	TURNING

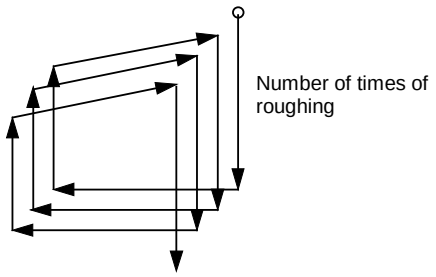
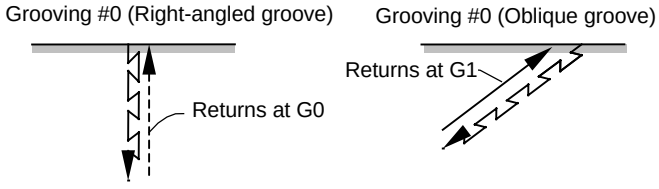
Address	Meaning		Description
TC63	Amount of relief after transfer using the **TRANSFER** unit (Spindle mode 0 to 5)		Set the amount of relief after executing the **TRANSFER** unit. **Example:** Transfer from HD2 to HD1 
	Program type	M	
	Conditions	After stop of movement	
	Unit	0.1 mm/0.01 inch	
	Setting range	0 to 65535	
TC64	Amount of relief after transfer using the **TRANSFER** unit (Spindle mode 6 and 7)		
	Program type	M	
	Conditions	After stop of movement	
	Unit	0.1 mm/0.01 inch	
	Setting range	0 to 65535	
TC65	Specification of first M-code for parts catcher control		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. **Note:** If 0 is set in **TC65** no M-code is outputted.
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 255	
TC66	Minimum index angle of the FLASH tool		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.)
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.1°	
	Setting range	0 to 3599	

Classification	USER	Display title	TURNING

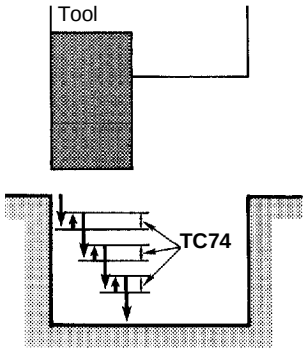
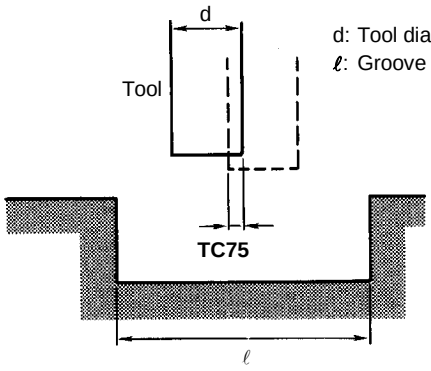
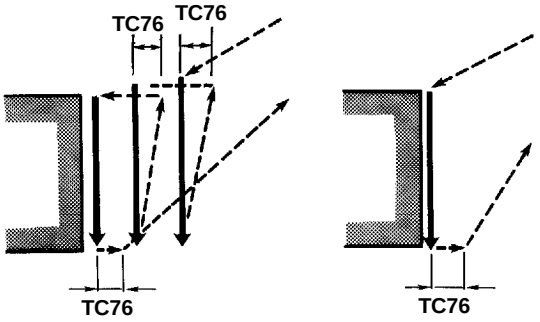
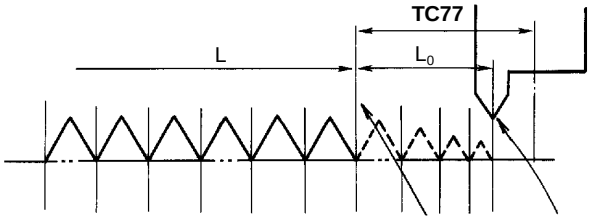
<table>
<tr><th>Address</th><th colspan="2">Meaning</th><th>Description</th></tr>
<tr><td rowspan="5">TC67</td><td colspan="2">Return distance (radial value) in X-axis at wall during rough cutting in bar cutting unit or in corner machining unit of MAZATROL programs</td><td rowspan="9">Example 1: Standard type cutting
Prior to cutting up along the wall in the end of final cycle, escape will be made by specified distance.
 

Example 2: High speed rough cutting
Escape will be made by TC67 and TC68 specified distance during return after reaching the wall.
 
</td></tr>
<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 inch</td></tr>
<tr><td>Setting range</td><td>0 to 65535</td></tr>
<tr><td rowspan="5">TC68</td><td colspan="2">Return distance in Z-axis at wall during rough cutting in bar cutting unit or in corner machining unit of MAZATROL programs</td></tr>
<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 inch</td></tr>
<tr><td>Setting range</td><td>0 to 65535</td></tr>
<tr><td rowspan="5">TC69</td><td colspan="2">Number of revolutions during dwell for pecking of grooving</td><td rowspan="5">Set number of revolutions during dwell for each cutting operation for machining #0, #1, #2, #3, #4, or #5 groove (GRV unit) with pecking return distance (TC74) of 0.
The tool stops moving until the spindle makes revolutions set in this parameter.</td></tr>
<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>
<tr><td rowspan="5">TC70</td><td colspan="2">FLASH tool
— Number of cutting edges to be used for the tool not registered in the tool file</td><td rowspan="5">Number of cutting edges to be used if the tool file data for the corresponding tool is not yet registered during FLASH tool selection.</td></tr>
<tr><td>Program type</td><td>M</td></tr>
<tr><td>Conditions</td><td>Immediate</td></tr>
<tr><td>Unit</td><td>Number of cutting edges</td></tr>
<tr><td>Setting range</td><td>0 to 99</td></tr>
</table>

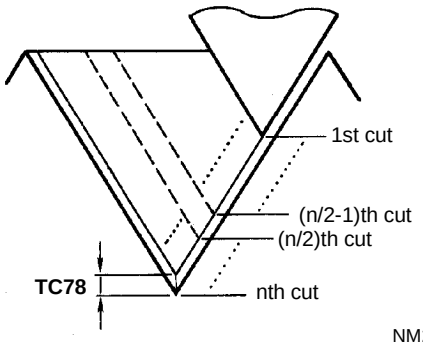
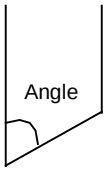
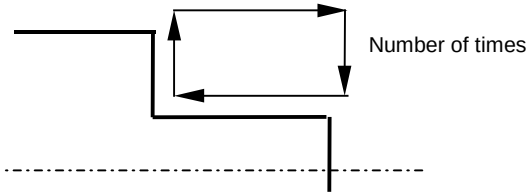
Classification	USER	Display title	TURNING

Address	Meaning		Description
TC71	Feed stopping rotation dwell time during the chip cutting cycle (valid only for roughing)		Specify the feed stopping rotation dwell time during the chip cutting cycle. Feed is stopped while the spindle rotates for the specified time. **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.
	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)		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. 
	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		Pecking speed [1] Cutting: (G1) F command data [2] Pecking: (G1) **TC73** [3] Cutting: (G1) F command data 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.  **Note:** In turning-drilling unit, **TC73** setting will be used only for #2 and [#2] types.
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/rev 0.0001 inch/rev	
	Setting range	0 to 65535	

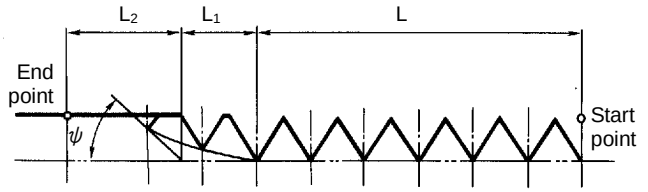
Classification	USER	Display title	TURNING

Address	Meaning		Description
TC74	Pecking return distance in groove cutting unit and grooving (G274/G275, G74/G75)		 NM211-00248
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 65535	
TC75	Overlap distance for machining wide groove in groove cutting unit		 d: Tool diameter ℓ: Groove width NM211-00249
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 65535	
TC76	Escape value after machining in edge machining unit		<Rough cutting> <Finish cutting>  NM211-00250
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 65535	
TC77	Acceleration distance clamp value for thread cutting unit		 L : Effective thread length L_0: Acceleration distance NM211-00246
	Program type	M	
	Conditions	Immediate	If L_0 > **TC77**, alarm will be caused. If, however, **TC141** (bit 2) = 0, alarm will not be caused.
	Unit	Lead/10	
	Setting range	0 to 255	(⇨**TC141** (bit 2))

Classification	USER	Display title	TURNING

Address	Meaning		Description
TC78	Cut depth (diametral value) for final cut in thread cutting unit Cut depth (diametral value) for final cut in composite-type thread cutting cycle G276		 NM211-00247
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 65535	
TC79	Minimum cut depth clamping value in thread cutting unit and composite-type thread cutting cycle G276		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. Clamping will follow the setting of **TC79** if the calculated value of the cut depth with the composite type thread cutting cycle G276 is smaller than the setting of **TC79**. This parameter is valid only for the infeed operation of the fixed volume chip production scheme.
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 65535	
TC80	Angle of the tool nose during the G276 mode		If, during the composite-type fixed cycle G276 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.
	Program type	E	
	Conditions	Immediate	
	Unit	1°	
	Setting range	0, 29, 30, 55, 60, 80	
TC81	Final finishing repeat times in the composite-type fixed cycle (G276)		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	Times	
	Setting range	0 to 65535	

Classification	USER	Display title	TURNING

Address	Meaning		Description
TC82	Chamfering data calculation coefficient in thread cutting unit and thread cutting cycle (G276/G292, G76/G92)		 NM211-00272 L : Effective thread length L_1 : Same pitch incomplete thread length (follow-up delay) L_2 : Chamfering data ψ : Chamfering angle $L_2 = L_0 \times \frac{\textbf{TC82}}{10}$ L_0 : Thread lead
	Program type	M, E	
	Conditions	Immediate	
	Unit	Lead/10	
	Setting range	0 to 40	
TC83	Number of cutting operations to be performed on finishing allowance corresponding to standard pattern (#0) of threading unit		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**. **TC83** = 0 or 1: Cutting based on the setting of **TC78** occurs once. **TC83** ≥ 2: Finish-cutting is repeated the number of times specified in **TC83**, and with the depth-of-cut setting of **TC78**/**TC83**. <Supplementary description> - Parameter **TC83** is valid only for #0, [#0]: it does not function for #1, [#1] or #2, [#2]. - If **TC78** = 0, **TC83** is valid. For thread refinishing, one cutting operation is performed as before.
	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		Specify the feed rate to be auto-set for finishing (turning) in the MAZATROL program.
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 65535	
TC85 to **TC94**	Specification of the pocket for the long boring bar		Specify the number of the magazine pocket holding the adapter for the long boring bar.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 960	
TC95 **TC96**	Fixed value		
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
TC97	Type of retraction during workpiece transfer		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.
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 2	

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

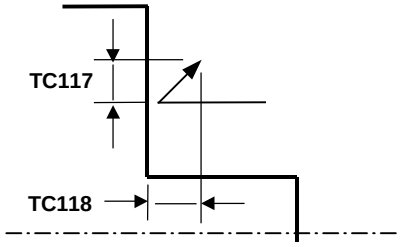
Classification	USER	Display title	TURNING

Address	Meaning		Description
TC98	Returning operation after machining specified in the END unit		Specify the sequence of the returning operation set in the **ATC** and **RETURN** items in the END unit for machines with the lower turret. **TC98** = 0: The upper and lower turrets return simultaneously. **TC98** = 1: The upper turret returns prior to the lower turret. **TC98** = 2: The lower turret returns prior to the upper turret. The setting is regarded as “0” if out of range.
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 2	
TC99	ATC operation after machining when not specified in the END unit		Specify the ATC operation after machining when the **ATC** item in the END unit is not set. **TC99** = 0: The same operation as when the item is set to “0” is followed. (The tool is not returned.) **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.) **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). The setting is regarded as “0” if out of range.
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 2	
TC100	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
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	

Classification	USER	Display title	TURNING

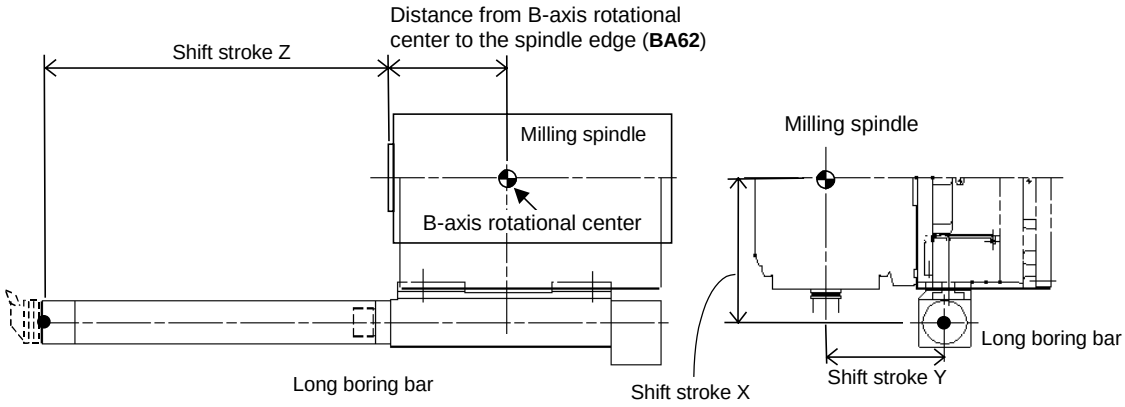
Address	Meaning		Description
TC102	Selection of cycle counter sampling axis (For detecting imbalance)		Select a cycle counter sampling axis.
	Program type	E	
	Conditions	At power on	
	Unit	—	
	Setting range	0 to 16	
TC103	Amplitude limit of table vibration (For detecting imbalance)		Set the amplitude limit of table vibration.
	Program type	E	
	Conditions	At power on	
	Unit	—	
	Setting range	0 to 9999	
TC104 to **TC110**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
TC111 to **TC113**	**CUTTING CONDITON LEARN** display — Workpiece length range		Specify the workpiece length range displayed on the **CUTTING CONDITON LEARN** display. Specify the range so that the relational expression of **TC111** < **TC112** < **TC113** is established.
	Program type	M	
	Conditions	Immediate	
	Unit	1 mm/0.1 inch	
	Setting range	0 to 65535	

Classification	USER	Display title	TURNING

Address	Meaning		Description
TC114 to **TC116**	**CUTTING CONDITON LEARN** display — Max. workpiece outside diameter range		Specify the maximum outside diameter range of the workpieces displayed on the **CUTTING CONDITON LEARN** display. Specify the range so that the relational expression of **TC114** < **TC115** < **TC116** is established.
	Program type	M	
	Conditions	Immediate	
	Unit	1 mm/0.1 inch	
	Setting range	0 to 65535	
TC117	Composite-type fixed cycle — G273/G73 amount of X-axial release		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 inch	
	Setting range	0 to 65535	
TC118	Composite-type fixed cycle — G273/G73 amount of Z-axial release		
	Program type	E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 65535	
TC119	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

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.



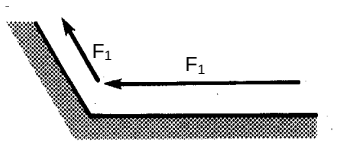
Address	Description
TC120	Long boring bar No. 1 - Shift stroke X
TC121	Long boring bar No. 1 - Shift stroke Y
TC122	Long boring bar No. 1 - Shift stroke Z
TC123	Long boring bar No. 2 - Shift stroke X
TC124	Long boring bar No. 2 - Shift stroke Y
TC125	Long boring bar No. 2 - Shift stroke Z
TC126	Long 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	Immediate	
Unit	Micron	0.001 mm 0.0001 inch
	Submicron for rotational axis	0.0001 mm 0.00001 inch
	Submicron for all axes	0.0001 mm 0.00001 inch
Setting range	±99999999	

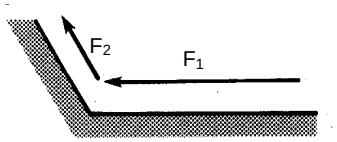
Address	Meaning	Description
TC138 to **TC140**	—	Invalid

Program type	—
Conditions	—
Unit	—
Setting range	—

Classification	USER	Display title	TURNING

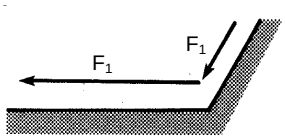
<table>
<tr><th>Address</th><th colspan="2">Meaning</th><th>Description</th></tr>
<tr><td rowspan="5">TC141
(bit 0)</td><td colspan="2" rowspan="5">Use/disuse of acceleration in up-going slope during rough cutting cycle in bar machining unit</td><td rowspan="5">TC141 (bit 0) = 0: Disuse
 

No acceleration

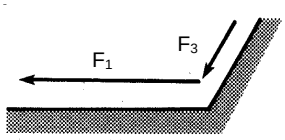
TC141 (bit 0) = 1: Use
 

$F_2 > F_1$
F_1 : Feed rate for rough cutting cycle
F_2 : Feed rate after acceleration

NM211-00208

(⇨TC2, TC3)</td></tr>
<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>
<tr><td rowspan="5">TC141
(bit 1)</td><td colspan="2" rowspan="5">Use/disuse of deceleration in down-going slope during rough cutting cycle in bar machining unit</td><td rowspan="5">TC141 (bit 1) = 0: Disuse
 

No deceleration

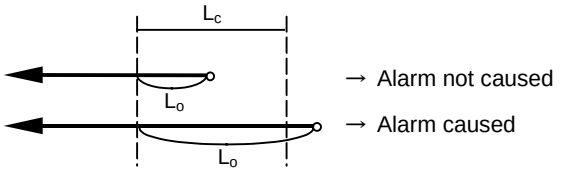
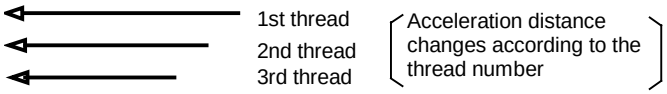
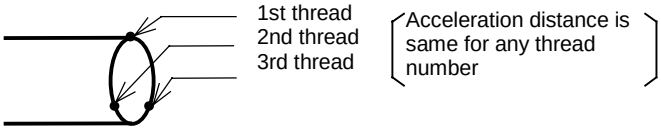
TC141 (bit 1) = 1: Use
 

$F_3 < F_1$
F_1 : Feed rate for rough cutting cycle
F_3 : Feed rate after deceleration

NM211-00209

(⇨TC5, TC6)</td></tr>
<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>

Classification	USER	Display title	TURNING

Address	Meaning		Description
TC141 (bit 2)	Selection between use/disuse of acceleration distance check at start of thread cutting unit		**TC141** (bit 2) = 0: Disuse Alarm not caused even if acceleration distance at start of thread cutting unit exceeds clamp data **TC141** (bit 2) = 1: Use Alarm caused when acceleration distance at start of thread cutting unit exceeds clamp data **Example:**  L_o : Acceleration distance L_c : Acceleration distance clamp data
	Program type	M	
	Conditions	Immediate	
	Unit	—	(⇨**TC77**)
	Setting range	0, 1	
TC141 (bit 3)	Selection between start position shift/start angle shift for thread number offset in thread cutting unit		**TC141** (bit 3) = 0: Start position shift Thread number offset adjusted by moving thread cutting start position  **TC141** (bit 3) = 1: Start angle shift Thread number offset adjusted according to thread cutting start angle 
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	

Classification	USER	Display title	TURNING

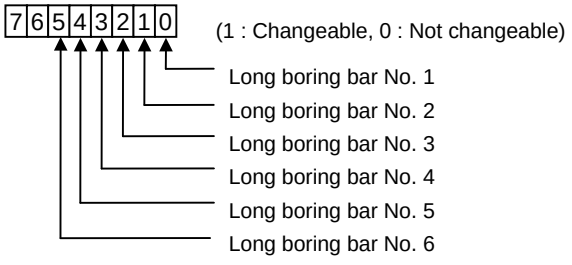
<table>
<tr><th>Address</th><th colspan="2">Meaning</th><th>Description</th></tr>
<tr><td rowspan="5">TC141
(bit 4)
(bit 5)</td><td colspan="2">Selecting an angle margin for nose shape compensation</td><td rowspan="5">An angle margin for nose shape compensation can be selected by setting data in bits 4 and 5.

<table><tr><th colspan="2">Setting</th><th rowspan="2">Angle margin for nose shape compensation</th></tr><tr><th>Bit 5</th><th>Bit 4</th></tr><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></table></td></tr>
<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>
<tr><td rowspan="5">TC141
(bit 6)</td><td colspan="2">CHUCK JAW DATA display name/code selection</td><td rowspan="5">TC141 (bit 6) = 1
A name can be selected.
TC141 (bit 6) = 0
A code can be selected.</td></tr>
<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>
<tr><td rowspan="5">TC141
(bit 7)</td><td colspan="2">Whether to make the partition plate and the workpiece barrier valid</td><td rowspan="5">TC141 (bit 7) = 1
Partition plate and the workpiece barrier valid
TC141 (bit 7) = 0
Partition plate and the workpiece barrier invalid</td></tr>
<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>
<tr><td rowspan="5">TC142
(bit 0)</td><td colspan="2">Fixed value (0)</td><td rowspan="5"></td></tr>
<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>

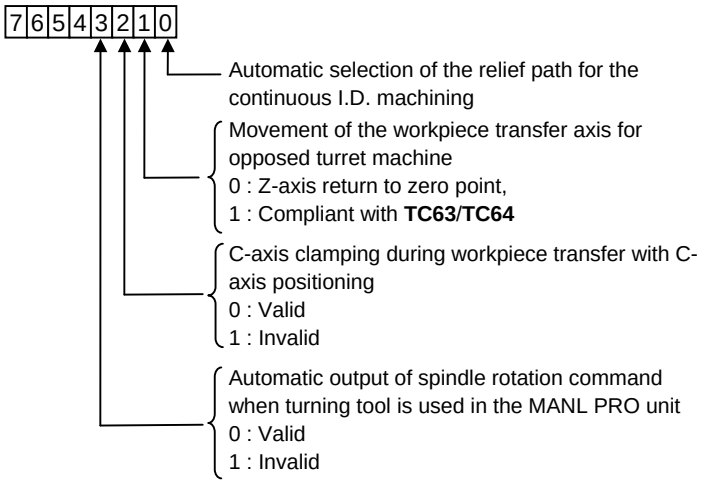
Classification	USER	Display title	TURNING

Address	Meaning		Description
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		**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. **TC142** (bit 1) = 1 The tool escapes to the clearance point each time the I.D. turning unit is executed to completion.
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
TC142 (bit 2)	Selection of the jaw data reference method		Specify the jaw data reference method. **TC142** (bit 2) = 0: Reference using the code number of the jaw. **TC142** (bit 2) = 1: Reference using the name of the jaw.
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
TC142 (bit 3)	Using angle tool holder valid/invalid		**TC142** (bit 3) = 0: Angle tool holder can not be used. **TC142** (bit 3) = 1: Angle tool holder can be used.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
TC142 (bit 4)	Selection of the method of moving axes to the tool change position		**TC142** (bit 4) = 0: The X-axis and the Z-axis simultaneously move to the tool change position specified by **F141** or **TC62**. **Note:** When the tool nose stays within the (workpiece diameter + safety profile clearance), both axes move past the clearance position. **TC142** (bit 4) = 1: The X-axis first and then the Z-axis move to the tool change position specified by **F141** or **TC62**. **Note:** When the tool nose stays within the (workpiece diameter + safety profile clearance), only the X-axis moves past the clearance position and the Z-axis does not move.
	Program type	M	
	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		**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. **TC142** (bit 5) = 1: An alarm will be issued if the ending position of workpiece pressing is reached during the transfer of the workpiece.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
TC142 (bit 6)	—		**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. **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.
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
TC142 (bit 7)	—		Upper turret retraction druing machining with the lower turret (for MULTIPLEX series) **TC142** (bit 7) = 0: Upper turret retraction to the X-axis zero point **TC142** (bit 7) = 1: Upper turret retraction invalid
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
TC143	Whether the end tool of the long boring bar can be changed		Whether the end tool of the long boring bar can be changed by ATC (1: Changeable, 0: Not changeable) 
	Program type	M, E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	Binary, eight digits	

Classification	USER	Display title	TURNING

Address	Meaning		Description
TC144	—		
	Program type	M	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	Binary, eight digits	
TC145 to **TC154**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

2-3-8 User parameter SOLID (SD)

Classification	USER	Display title	SOLID

Address	Meaning		Description
SD1 to **SD48**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
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	
SD50	Table type		Select a table type for 3D setup display. 0: Square table 1: Round table 2: Chuck
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 2	
SD51 to **SD96**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

Classification	USER

Display title	SOLID

Address	Meaning		Description
SD97	Distance of the model movement per time		Set the default value for the model's movement distance per time that is to be displayed in the movement distance assignment dialog box.
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.0001 mm/0.0000I inch	
	Setting range	0 to 99999999	
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	
SD99 to **SD124**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

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)	J51 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)	—	—	—	—	Invalid

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

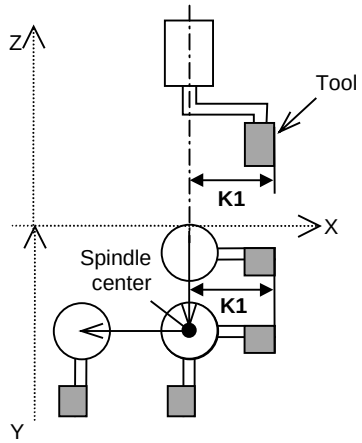
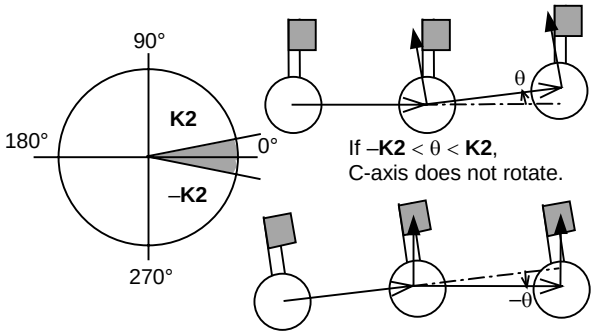
Classification	MACHINE

Display title	CALL MACRO

Address	Meaning		Description
J81 to **J90**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
J91 to **J107**	—		Parameter for system internal setting Setting prohibited
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
J108 to **J144**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

2-3-10 Machine parameter MEASURE (K)

Classification	MACHINE	Display title	MEASURE

Address	Meaning		Description
K1	Rotational radius of the C-axis		Distance from the center of the C-axis (spindle) to the nose of the tool 
	Program type	E	
	Conditions	Immediate	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to 99999999	
K2	Minimum rotational angle		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 l θ l < **K2**. 
	Program type	E	
	Conditions	Immediate	
	Unit	0.001 deg	
	Setting range	0 to 90000	

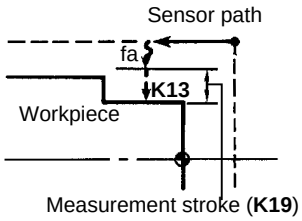
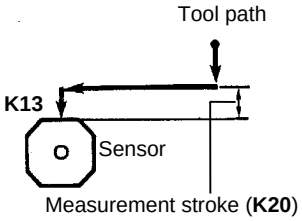
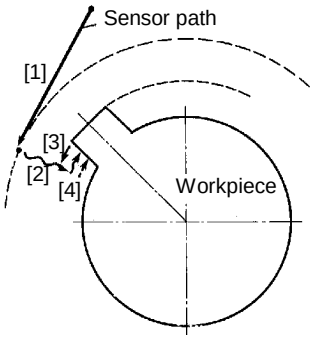
Classification	MACHINE	Display title	MEASURE

Address	Meaning		Description
K3	Shaping control axis		Identification number of the shaping control axis Specify the shaping control axis as follows: **Example:** Set “4” for a three-axis machine. Set “5” for a four-axis machine. **Note:** Set the type of the axis which has been set on this parameter to the rotational axis (**M17** bit 4 = 1).
	Program type	E	
	Conditions	Immediate	
	Unit	Axis	
	Setting range	3 to 13	
K4 to **K6**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
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.
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	1, 2, 4	
K8 **K9**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

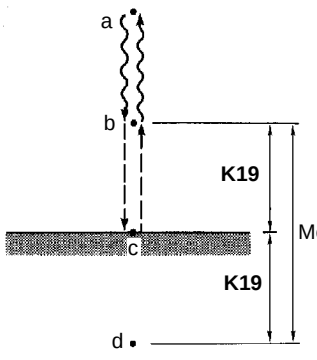
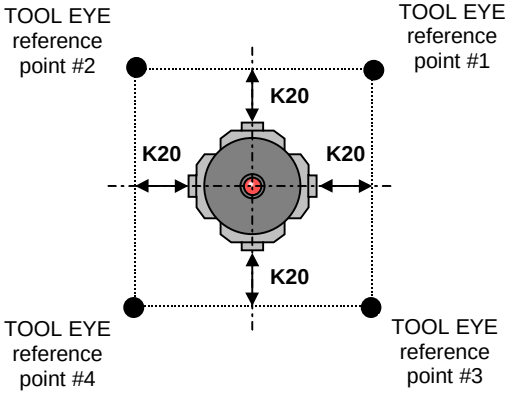
Classification	MACHINE	Display title	MEASURE

<table>
<tr><th>Address</th><th colspan="2">Meaning</th><th>Description</th></tr>
<tr><td rowspan="5">K10</td><td colspan="2">Fixed value (0)</td><td rowspan="5"></td></tr>
<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>
<tr><td rowspan="5">K11</td><td colspan="2">Selection of language to be displayed</td><td rowspan="5">Set this parameter to change the display language.
<table><tr><th>Setting</th><th>Language</th><th>Setting</th><th>Language</th></tr><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></table>Note:
To display Japanese characters, traditional and/or simplified Chinese characters, Korean characters, or Russian characters, the appropriate OS (display language) must be installed in your personal computer.</td></tr>
<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 21</td></tr>
<tr><td rowspan="5">K12</td><td colspan="2">Fixed value (0)</td><td rowspan="5"></td></tr>
<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>

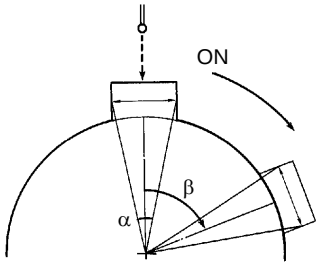
Classification	MACHINE	Display title	MEASURE

Address	Meaning		Description
K13	Measurement skip feed rate (X-axis, Z-axis)		(1) Workpiece measurement  fa: Measurement approach speed (**K14**) NM211-00233 (2) Tool tip measurement  NM211-00234 (⇨**K19**, **K20**)
	Program type	M	
	Conditions	Immediate	
	Unit	1 mm/min / 0.1 inch/min	
	Setting range	0 to 20000	
K14	Measurement approach speed (X-axis, Z-axis)		
	Program type	M	
	Conditions	Immediate	
	Unit	1 mm/min / 0.1 inch/min	
	Setting range	0 to 20000	
K15	Measurement skip speed (C-axis)		 [1], [3] : Rapid feed [2] : Measurement approach speed (**K16**) [4] : **K15** NM211-00235
	Program type	M	
	Conditions	Immediate	
	Unit	1°/min	
	Setting range	0 to 65535	
K16	Measurement approach speed (C-axis)		
	Program type	M	
	Conditions	Immediate	
	Unit	1°/min	
	Setting range	0 to 65535	

Classification	MACHINE	Display title	MEASURE

Address	Meaning		Description
K17	Specification of measuring tolerance (lower limit)		(1) Tool compensation will be made in cases below. $\frac{a_1 - a_2}{100} \times$ **K18** $\geq$ Compensation data $\geq \frac{a_1 - a_2}{100} \times$ **K17** $-\frac{a_1 - a_2}{100} \times$ **K17** $\geq$ Compensation data $\geq -\frac{a_1 - a_2}{100} \times$ **K18** (2) Tool compensation will not be made in cases below. $\frac{a_1 - a_2}{100} \times$ **K17** > Compensation data > $-\frac{a_1 - a_2}{100} \times$ **K17** (3) Alarm will be caused in cases below. Compensation data > $\frac{a_1 - a_2}{100} \times$ **K18** $-\frac{a_1 - a_2}{100} \times$ **K18** > Compensation data where a_1 = Tolerance upper limit a_2 = Tolerance lower limit (Specified in program) **Note:** **1.** Up to 100 % can be set. **2.** Offset judgement occurs only when **L106** bit 6 ist set to 1.
	Program type	M, E	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 100	
K18	Specification of measuring tolerance (upper limit)		
	Program type	M, E	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 100	
K19	Measurement stroke for workpiece measurement		 a : Approach point b : Measurement start point c : Measurement point (target data) d : Measurement end point NM211-00259
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 65535	
K20	Measurement stroke for tool nose measurement		
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	±99999999	

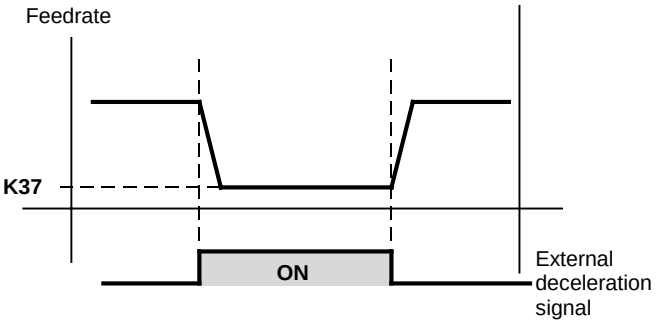
Classification	MACHINE	Display title	MEASURE

Address	Meaning		Description
K21	Coefficient to determine rotation angle when retrying measurement C reference face		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**. **Example:** CW programmed as measurement direction  α : Measurement object shape angle β : α × **K21**/100 C-axis rotation by angle β in this direction NM211-00261
	Program type	M	
	Conditions	Immediate	
	Unit	%	
	Setting range	1 to 1000	
K22	Measurement retry frequency when retrying reference face C measurement		When touch sensor is actuated before reaching target point in C offset measurement, set retry frequency. 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.
	Program type	M	
	Conditions	Immediate	
	Unit	Times	
	Setting range	0 to 255	
K23	Retry frequency for workpiece measurement		Specify the number of times the workpiece measurement is to be retried if a measurement error occurs.
	Program type	M	
	Conditions	Immediate	
	Unit	Times	
	Setting range	0 to 99999	
K24 to **K28**	Fixed value		Reserved within the system. Setting prohibited
	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		1. Delays master/slave axis position matching during automatic error correction (parameter **K103** bit 1 = 0). 2. Delays pitch error correction data output when the servo is ON. **Note:** The delay time will be 3 seconds if 0 is set.
	Program type	M, E	
	Conditions	Immediate	
	Unit	× 1.7 msec	
	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	1 mm/min / 0.1 inch/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	—		Reserved within the system. Setting prohibited
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

Classification	MACHINE	Display title	MEASURE

Address	Meaning		Description
K37	External deceleration speed		The upper-limit value of the feed rates available while the external deceleration signal is ON  MPL508
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	1 mm/min	
	Setting range	0 to 120000	
K38	Work number called during S-code macroprogram appointment		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.) **Note:** This parameter is valid only when bit 2 of parameter **K105** is 1.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 999999999	
K39	Work number called during T-code macroprogram appointment		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.) **Note:** This parameter is valid only when bit 3 of parameter **K105** is 1.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 999999999	
K40	Work number called during second auxiliary function macroprogram appointment		The work number of the macroprogram to be called during macroprogram appointment using the second auxiliary function **Notes:** **3.** This parameter is valid only when bit 4 of parameter **K105** is 1. **4.** 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	

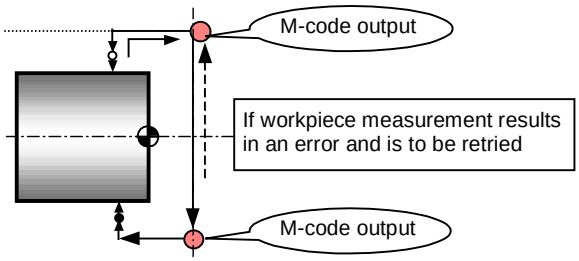
Classification	MACHINE	Display title	MEASURE

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	1 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	1 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	1 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	1 mm/min	
	Setting range	0 to 120000	

Classification	MACHINE	Display title	MEASURE

Address	Meaning		Description
K45	G31.4 skipping speed		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.
	Program type	E	
	Conditions	After stop of movement	
	Unit	1 mm/min	
	Setting range	0 to 120000	
K46	Excessive pressing error spread (Amount of drooping)		Set the excessive pressing error spread (the amount of drooping).
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.0001 mm	
	Setting range	0 to 32767	
K47	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
K48	Spindle FB alarm detection method		0 : Monitoring based on acceleration 1 : Monitoring based on speed 2 : Ignoring alarms
	Program type	M, E	
	Conditions	Immediate	
	Unit		
	Setting range	0 to 2	

Classification	MACHINE	Display title	MEASURE

Address	Meaning		Description
K49	First number of the standby M-codes		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**.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	—	
	Setting range	31 to 1000	
K50	Total number of the standby M-codes		Set the total number of 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**. **Note:** If “0” is set, the M-code standby function will be invalid.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	—	
	Setting range	0 to 1000	
K51	M-code during workpiece measurement retry operation		Set the M-code to be output if workpiece measurement results in an error and is to be retried. **Note:** If “0” is set, this function will be invalid. 
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	—	
	Setting range	0 to 1000	
K52	—		Parameter for system internal setting Setting prohibited
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

Classification	MACHINE	Display title	MEASURE

Address	Meaning		Description
K53	Language selection (vocal output)		Set the type of vocal output language. Setting / Language / Setting / Language 0 English — 8 — 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 — — — **Note:** Output will be in English (0) if the selected value is for a language not supported.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 21	
K54	Sound level (vocal output)		Set the vocal output sound level. **Note:** Entry of “0” means no sound level.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 100	
K55	Warning reference value (vocal output)		A vocal warning will be output if the value of the 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: Address name / Setting (HEX) Invalid 0 A 41 B 42 C 43 **Note:** Do not use the same address for the axis name and the second auxiliary function.
	Program type	E	
	Conditions	At power on	
	Unit	—	
	Setting range	Hexadecimal two-digit 0, 41, 42, 43	

Classification	MACHINE	Display title	MEASURE

Address	Meaning		Description
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. Setting: 0 — Calling method: M98 P□□□□ Setting: 1 — Calling method: G65 P□□□□ Setting: 2 — Calling method: G66 P□□□□ Setting: 3 — Calling method: G66.1 P□□□□ **Note:** Valid only when bit 2 of **K105** is 1.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 3	
K58	Type of T-code macroprogram appointement 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. Setting: 0 — Calling method: M98 P□□□□ Setting: 1 — Calling method: G65 P□□□□ Setting: 2 — Calling method: G66 P□□□□ Setting: 3 — Calling method: G66.1 P□□□□ **Note:** Valid only when bit 3 of **K105** is 1.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 3	
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 macroporgram whose work number has been set using the **K40** parameter. Setting: 0 — Calling method: M98 P□□□□ Setting: 1 — Calling method: G65 P□□□□ Setting: 2 — Calling method: G66 P□□□□ Setting: 3 — Calling method: G66.1 P□□□□ **Note:** Valid only when bit 4 of **K105** is 1.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 3	
K60	Fixed value (4)		
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

Classification	MACHINE

Display title	MEASURE

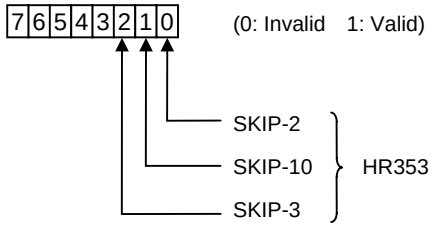
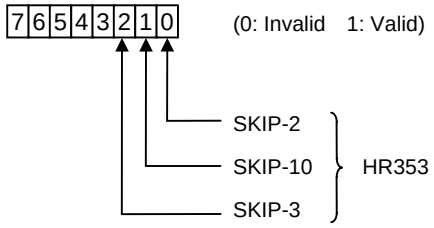
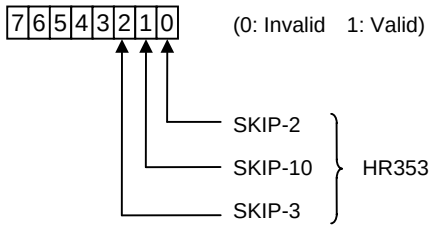
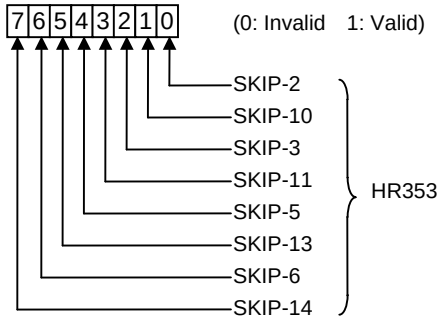
Address	Meaning		Description
K61	Fixed value (1)		
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
K62	Fixed value (1)		
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
K63	Fixed value (1)		
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
K64	Fixed value (2)		
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

Classification	MACHINE

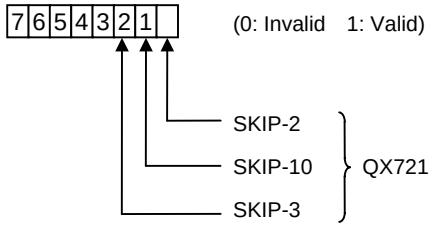
Display title	MEASURE

Address	Meaning		Description
K65	Fixed value (1)		
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
K66	Fixed value (1)		
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
K67	Fixed value (1)		
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
K68	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

Classification	MACHINE	Display title	MEASURE

Address	Meaning		Description
K69	G31.1 skip conditions		
	Program type	E	Select the skip signal for G31.1 command.
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	Binary, eight digits	
K70	G31.2 skip conditions		
	Program type	E	Select the skip signal for G31.2 command.
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	Binary, eight digits	
K71	G31.3 skip conditions		
	Program type	E	Select the skip signal for G31.3 command.
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	Binary, eight digits	
K72	G31.4 skip conditions		
	Program type	E	Select the skip signal for the G31.4 command.
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	Binary, eight digits	

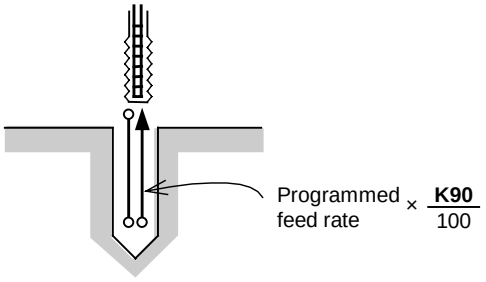
Classification	MACHINE	Display title	MEASURE

Address	Meaning		Description
K73	G4 skip conditions		 Select the skip signal for G4 command.
	Program type	E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	Binary, eight digits	
K74	Emergency stop contactor cutoff time (Safety supervisory function)		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. A contactor cutoff signal will be immediately output if all-axis zero-speed confirmation is executable earlier than the set time.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	sec	
	Setting range	0 to 60	
K75	Contactor control output device 1 (Safety supervisory function)		Specify a remote I/O device that is to output a contactor activating/deactivating signal when the safety supervisory function is executed. **Note:** The signal will not be output if “&0” is specified.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	—	
	Setting range	&0 to &7F	
K76	Contactor control output device 2 (Safety supervisory function)		Specify another remote I/O device that is to output the contactor activating/deactivating signal when the safety supervisory function is executed. **Note:** The signal will not be output if “&0” is specified.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	—	
	Setting range	&0 to &7F	

Classification	MACHINE	Display title	MEASURE

Address	Meaning		Description
K77	Door switch input device (Safety supervisory function)		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). **Note:** Door switch input will be invalid if “&0” is entered.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	—	
	Setting range	&0 to &7F	
K78	Number of door switches (Safety supervisory function)		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	
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 msec.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	1.777 msec	
	Setting range	0 to 1000	
K80 to **K84**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

Classification	MACHINE	Display title	MEASURE

Address	Meaning		Description
K85	Special linear acceleration/deceleration time constant for threading		If the setting of **K85** is from 1 to 3 msec, this setting will be used as the linear acceleration/deceleration time constant for the G32 threading block. If the setting is outside the valid range, however, the normal linear acceleration/deceleration time constant for G01 will be used.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	msec	
	Setting range	0 to 300	
K86 to **K89**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
K90	Return override during synchronous tapping		The overriding value for return from the hole bottom during a synchronous tapping cycle  MPL509 **Note:** This parameter is valid only when bit 6 of **F94** is 1.
	Program type	E	
	Conditions	After stop of movement	
	Unit	1%	
	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	

Classification	MACHINE	Display title	MEASURE

Address	Meaning		Description
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	—	
K94	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

Classification	MACHINE	Display title	MEASURE

Address	Meaning		Description
K95	—		7 6 5 4 3 2 1 0 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
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	Binary, eight digits	
K96	—		7 6 5 4 3 2 1 0 G0 command in-position check 0: Check 1: Non-check Timing of manual free feed finish signal 0: Smoothing 1: Distribution finish (equivalent to DEN) Fixed value (0) Axis/Cutting interlock alarm display 0: Valid 1: Invalid Suppression of lost motion in modes other than the G1 command mode 0: Valid 1: Invalid Fixed value (0) Fixed value (0) Fixed G0 inclination
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	Binary, eight digits	

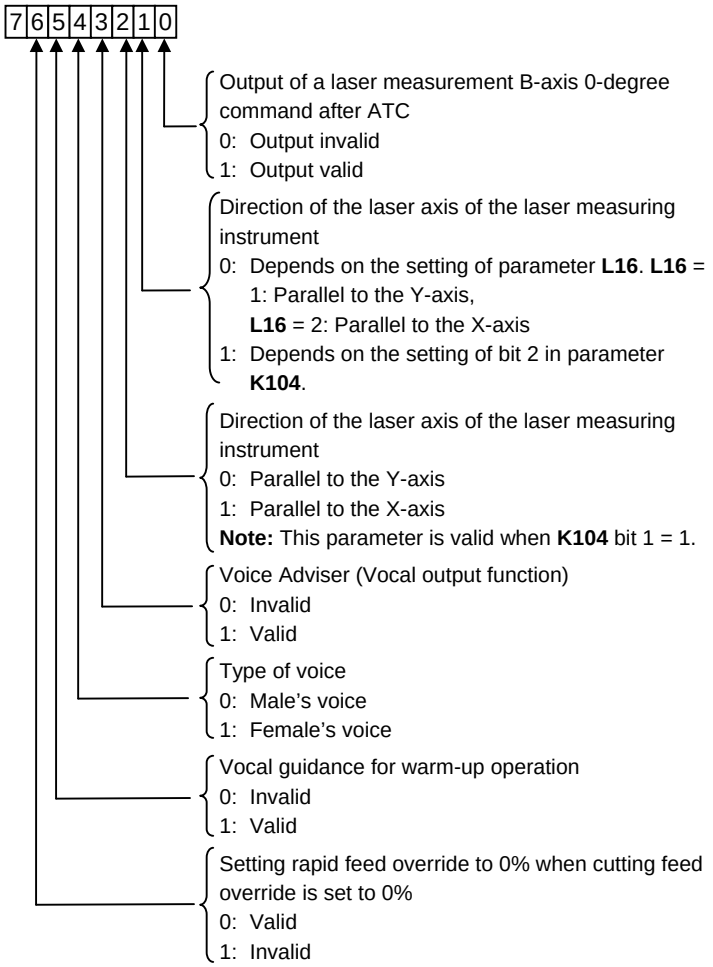
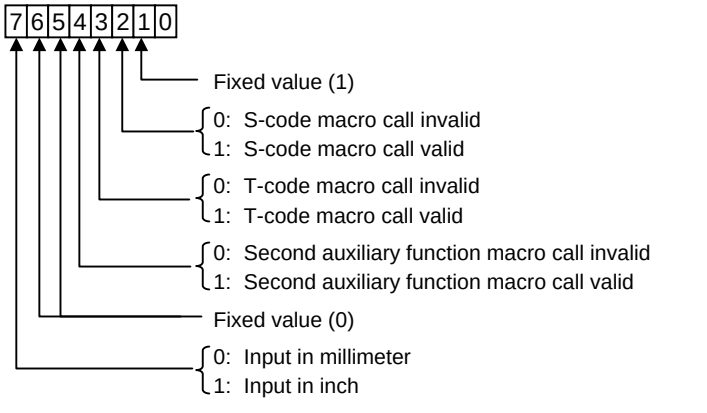
Classification	MACHINE	Display title	MEASURE

Address	Meaning		Description
K97	B-axis misalignment correction Name of parallel axis		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.
	Program type	M, E	
	Conditions	Immediate	
	Unit	Hexadecimal, two digits	**Note:** If 0 is entered, the axis will be regarded as the Z-axis (&5A).
	Setting range	&41 to &5A	
K98	B-axis misalignment correction Name of orthogonal axis		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.
	Program type	M, E	
	Conditions	Immediate	
	Unit	Hexadecimal, two digits	**Note:** If 0 is entered, the axis will be regarded as the X-axis (&58).
	Setting range	&41 to &5A	
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	

Classification	MACHINE	Display title	MEASURE

Address	Meaning		Description
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	Fixed value (0)		
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
K103	—		7 6 5 4 3 2 1 0 Synchronous control error auto-correction 0: Invalid 1: Valid DDB micron unit 0: Interpolation 1: Microns Thermal displacement compensation interval 0: 0.5 μm 1: Unit of compensation Output of S-code and T-code at restart 0: Invalid 1: Valid
	Program type	M, E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	Binary, eight digits	

Classification	MACHINE	Display title	MEASURE

Address	Meaning		Description
K104	Laser tool length/diameter measurement		
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
K105	—		
	Program type	M, E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	Binary, eight digits	

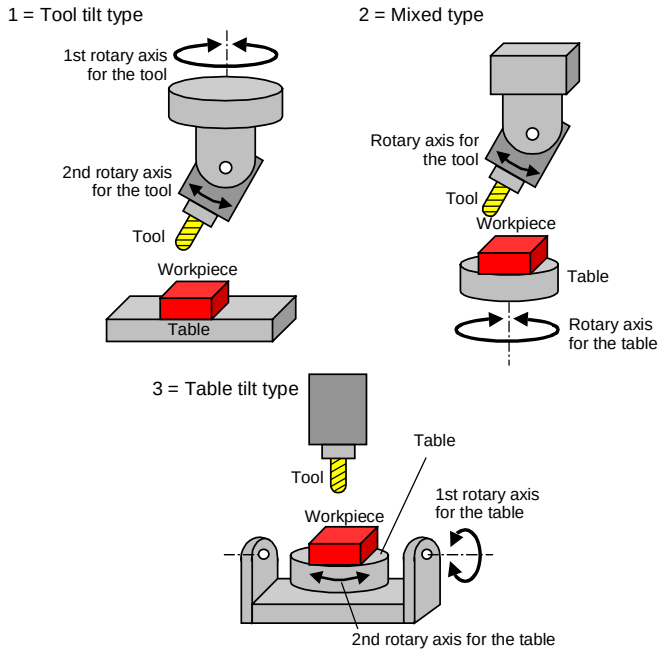
Classification	MACHINE	Display title	MEASURE

<table>
<tr><th>Address</th><th colspan="2">Meaning</th><th>Description</th></tr>
<tr><td rowspan="5">K106</td><td colspan="2">—</td><td rowspan="5">7 6 5 4 3 2 1 0
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)
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.
Fixed value (0)
Pitch error setting
0: Absolute
1: Incremental
Fixed value (0)
Fixed value (0)
Fixed value (0)
Fixed value (0)</td></tr>
<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>
<tr><td rowspan="5">K107</td><td colspan="2">—</td><td rowspan="5">7 6 5 4 3 2 1 0
Fixed value (0)
Deceleration for arc valid/invalid
Fixed value (0)</td></tr>
<tr><td>Program type</td><td>—</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>
<tr><td rowspan="5">K108</td><td colspan="2">Permissible error range for synchronous control</td><td rowspan="5">Specify the maximum permissible error range for the master and slave axes for synchronous control (Tandem driving system).
If the specified range is overstepped, the alarm EXCESS SIMULTANEOUS ERROR will be displayed.
Note:
Error checking will not occur if 0 is set.</td></tr>
<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>0 to 65535</td></tr>
</table>

Classification	MACHINE	Display title	MEASURE

Address	Meaning		Description
K109	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
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 deg.
	Program type	E	
	Conditions	After stop of axis movement	
	Unit	deg	
	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	
K112	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

Classification	MACHINE	Display title	MEASURE

Address	Meaning		Description
K113	Machine type (Tool tip point control)		Set the type of machine. 1 : Tool tilt type 2 : Mixed type 3 : Table tilt type **Example:** 
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	1 to 3	
K114	Axis number of the horizontal axis in the rectangular coordinate system (Tool tip point control)		Set the axis number of the horizontal axis in the rectangular coordinate system. A setting of 0 is invalid.
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 16	
K115	Axis number of the vertical axis in the rectangular coordinate system (Tool tip point control)		Set the axis number of the vertical axis in the rectangular coordinate system. 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
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 vertical axis. 32: 2nd rotary axis ..... Rotates about height axis, 1st rotary axis ..... Rotates about horizontal axis.
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	—	
K118 to **K120**	—		Invalid
	Program type	—	
	Conditions	—	
	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	

Classification	MACHINE	Display title	MEASURE

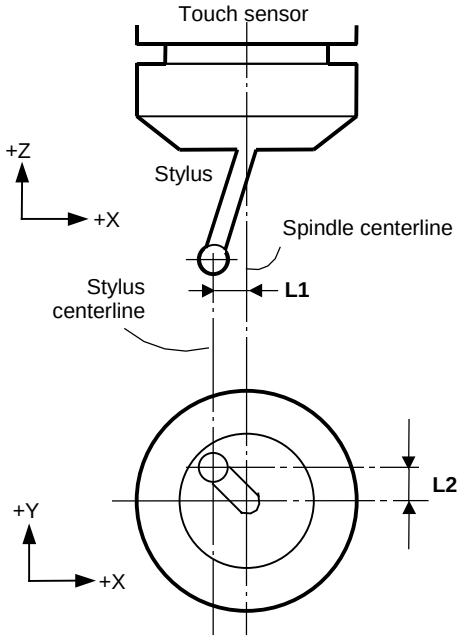
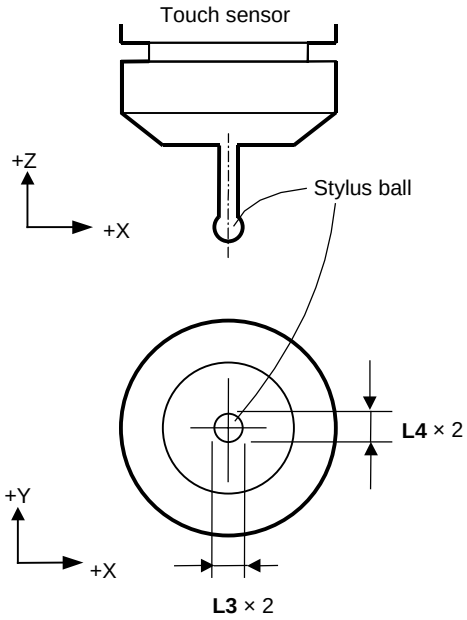
Address	Meaning		Description
K122	Horizontal axis rotational center offset of the first rotary axis (Tool tip point control)		<If the first rotary axis rotates for tool control> Set the distance from the rotational center of the tool control rotary axis (at the tool side) in the direction of the horizontal 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	
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). <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.
	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

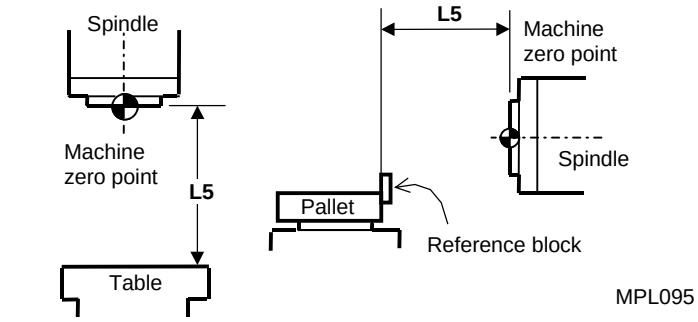
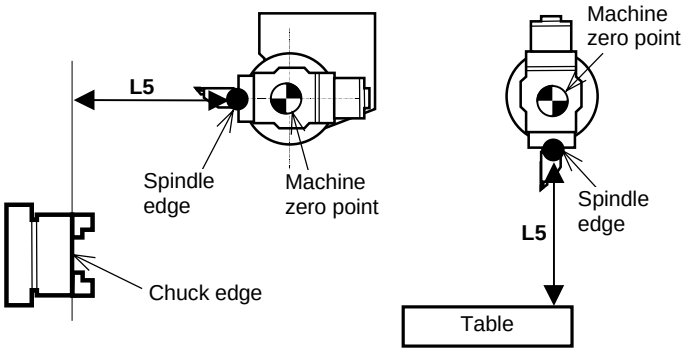
Address	Meaning		Description
K126	Horizontal axis rotational center offset of the second rotary axis (Tool tip point control)		<If the second rotary axis rotates for tool control> Set the distance from the spindle holder end in the direction of the horizontal axis to the rotational center of the tool 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	
K129 to **K144**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

2-3-11 Machine parameter TABLE (L)

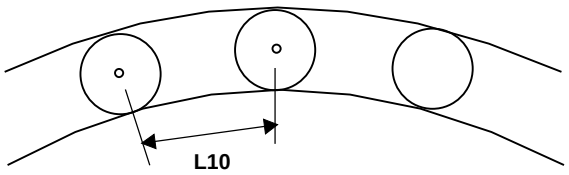
Classification	MACHINE	Display title	TABLE

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  MPL093
	Program type	M	
	Conditions	At power on	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to ±99999999	
L2	Stylus eccentricity of touch sensor (Y-component)		**Note:** These data are automatically set when calibration measurement is performed using the **MMS** unit.
	Program type	M	
	Conditions	At power on	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to ±99999999	
L3	Radius of stylus ball of touch sensor (X-component)		The true radius value of the stylus ball of the touch sensor  MPL094
	Program type	M	
	Conditions	At power on	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to ±99999999	
L4	Radius of stylus ball of touch sensor (Y-component)		**Note:** These data are automatically set when calibration measurement is performed using the **MMS** unit.
	Program type	M	
	Conditions	At power on	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to ±99999999	

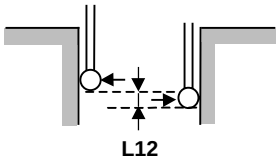
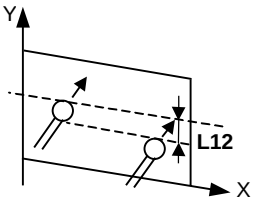
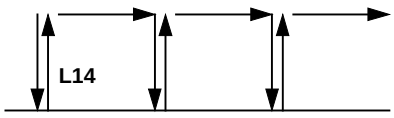
Classification	MACHINE	Display title	TABLE

Address	Meaning		Description
L5	Z-axis stroke for tip position memory (TEACH function)		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  MPL095  (Model H, e type) (Model V, e type)
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to ±99999999	
L6	Tool-breakage judgment distance for TBR function		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 If (registered tool length data) – (tool length data that has been measured during the detecting operation) ≥ **L6**, then the tool is judged broken.
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to ±99999999	
L7	Tool-breakage restoration mode for TBR function		The parameter for selecting the type of restoration to be performed after tool breakage has been detected as a result of execution of the tool breakage detection function 1: Single-block stop 2: Machining restarts from the next process. 3: Single-block stop occurs in a state where machining can be restarted from the next process.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	1 to 3	

Classification	MACHINE	Display title	TABLE

Address	Meaning		Description
L8	Skipping stroke limit for MMS		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.
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to 99999999	
L9	Selection of random ATC specifications		Set to 1 when the machine of the random ATC specifications is used. 0: Standard machine 1: Machine of random ATC specifications
	Program type	M, E	
	Conditions	At power on	
	Unit	—	
	Setting range	0, 1	
L10	Interval between magazine pockets		Set the interval between magazine pockets.  MPL511
	Program type	M, E	
	Conditions	Immediate	
	Unit	1 mm/0.1 inch	
	Setting range	0 to 999	
L11	Touch sensor’s interference direction		Set the touch sensor's interference direction. 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  MPL512
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to ±99999999	
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. **Note:** When error angle is smaller than the setting angle, the parallelism and right angle are calculated.
	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.  MPL513
	Program type	M	
	Conditions	Immediate	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to ±99999999	

Classification	MACHINE	Display title	TABLE

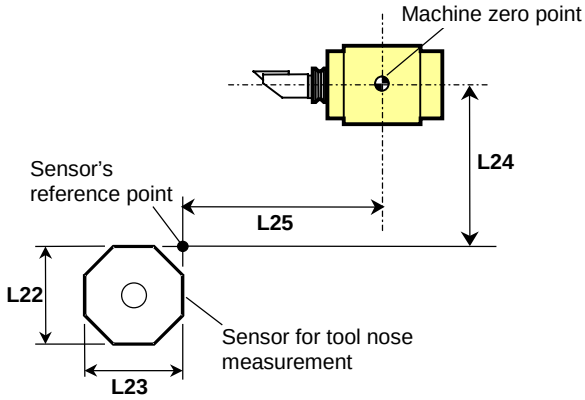
Address	Meaning		Description
L15	Macro program number for straightness measurement		Set macro program number for straightness measurement. Before shipment, the macro program is numbered “9999” at the factory.
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 99999999	
L16	Fixed value		Parameter for system internal setting Setting prohibited
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
L17	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
L18	Presence/absence of tailstock		0: Tailstock absent. 1: Tailstock present.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	

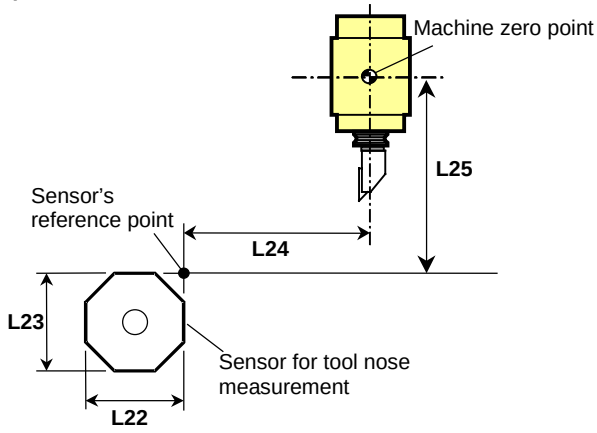
Classification	MACHINE

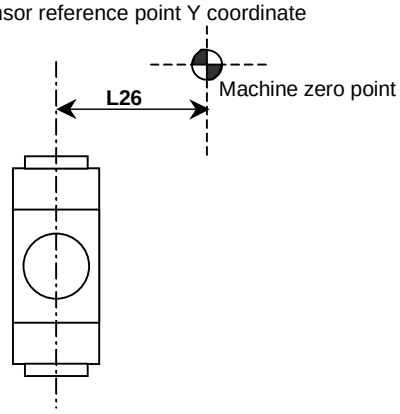
Display title	TABLE

Address	Meaning		Description
L19 **L20**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
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	

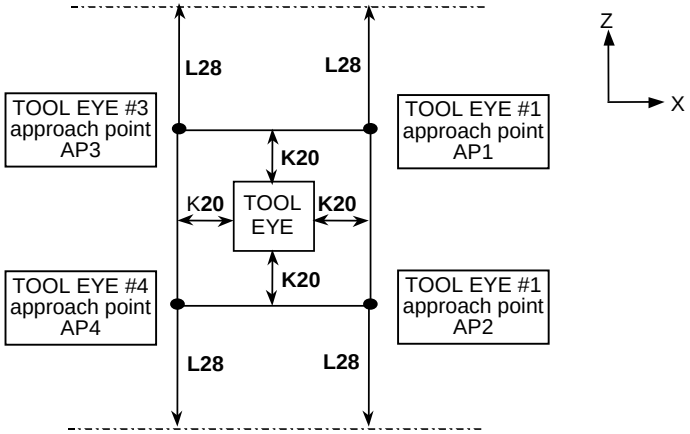
Classification	MACHINE	Display title	TABLE

<table>
<tr><th>Address</th><th>Meaning</th><th>Description</th></tr>
<tr><td rowspan="5">L22
to
L25</td><td>Data of the tool nose measurement sensor
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</td><td rowspan="5">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).
[INTE IV, INTE e-HII, lathes]
Example 1: For No. 1 turning spindle
 

Set the imaginary upper left corner of the sensor as its reference point for No. 2 spindle.
[INTE e-VII]
Example 2:
 

Note:
As shown above, the X- and Z-axes must be replaced with each other between INTEGREX e-HII and e-VII.</td></tr>
<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 inch</td></tr>
<tr><td>Setting range</td><td>±99999999</td></tr>
<tr><td rowspan="5">L26</td><td>Tool nose measurement sensor reference position, Y-axis</td><td rowspan="5">Setting of sensor reference point Y coordinate
 
</td></tr>
<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 inch</td></tr>
<tr><td>Setting range</td><td>±99999999</td></tr>
</table>

Classification	MACHINE	Display title	TABLE

Address	Meaning		Description
L27	Timer setting for manual TOOL EYE measurement		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.
	Program type	M, E	
	Conditions	Immediate	
	Unit	3.5 msec	
	Setting range	0 to 65535	
L28	Amount of Z-axial escape from the approach point after TOOL EYE measurement		Under the machine configuration where, after automatic tool measurement with the TOOL EYE, the cover of the TOOL EYE will interfere with the tool if the cover is closed with the tool present at the measuring approach point, enter the distance through which the tool is to be moved in the Z-axial direction before the cover is closed following completion of the measurement. In the case of MAZATROL programs, when the measurement is completed, the tool will be moved through the **L28**-specified distance from the approach point before the cover is closed. In the case of EIA/ISO programs, when execution of the G136 command is completed, the tool will be moved through the **L28**-specified distance from the approach point. The cover will be closed by execution of M284 in the next block onward. 
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	±99999999	
L29	Machine efficiency		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%.
	Program type	M	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 100	

Classification	MACHINE	Display title	TABLE

Address	Meaning		Description
L30	Selection of machining navigation case introduction messages		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.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 999	
L31 to **L36**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
L37	Minimum index angle of index table		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.
	Program type	M	
	Conditions	At power on	
	Unit	1°	
	Setting range	0 to 180	
L38	M/B-code for index of index table		For the command to rotate the index table, select the turning direction and the M-code number to be output. 0 or 1: B-code (0: Turning in the direction of CW, 1: Turning in the direction of CW/CCW/shortcut) 2 to 9999: M-code (Numeric value is the M-code number to be output) **Notes:** **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	

Classification	MACHINE	Display title	TABLE

Address	Meaning		Description
L39	Selection of execution/non execution of indexing unit		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: 0: Won't be made when the index angle is the same as the preceding indexing unit execution. 1: Will be made unconditionally.
	Program type	M	
	Conditions	At power on	
	Unit	—	
	Setting range	0, 1	
L40	Availability of specification of index table angle in end unit		Select availability to specify the index table angle in the end unit. 0: Enables to specify the angle. 1: Prohibits to specify the angle. **Note:** Set to 0 only for index table specification.
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
L41	Simultaneous operation of indexing unit with ATC		For execution of the indexing unit, specify the commanding order for movement to turning position, turning of the table and ATC. 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. **Note:** In case of setting to 2, only the X-axis coordinates can be set at turning position of the indexing unit.
	Program type	M	
	Conditions	At power on	
	Unit	—	
	Setting range	0 to 2	
L42	Initial value of index table angle		Select setting of the initial value (modal) of the index table angle for cyclic operation. 0: Actual table angle of the machine 1: Table angle indexed at present taken as 0°
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	

Classification	MACHINE	Display title	TABLE

Address	Meaning		Description
L43	Indication of index table angle		Select showing or not showing of the index table angle on the **POSITION** display. 0: Not to show 1: To show
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	**Note:** Set to 1 for the machine with the index table, or set to 0 for that of the NC rotary table.
	Setting range	0, 1	
L44	Selection of automatic setting on/off for nose position correction of a drilling tool		Select whether automatic setting of the amount of tool nose position correction is to be made valid or invalid when entering the length of a drilling tool in the tool data or when measuring the tool length in the MDI mode. 0: Automatic setting valid 1: Automatic setting invalid
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
L45	Index table angle command		Set the minimum unit of index table angle command for **INDEX** units, **ANGLE** in **END** units and B-codes in **MANL PRG** unit. 0: 1-deg 1 to 8: 1/1000 deg (MRJ2-CT specifications)
	Program type	—	**Note:** Index table angle display on the **POSITION** display is valid only when **L43** = 1 (index table angle display on). 0: 1-deg index table 1 to 7: Nth axis under MRJ2-CT specs. (N = 1 to 7) 8: Positioning table
	Conditions	—	
	Unit	—	
	Setting range	0 to 8	
L46	Maximum number of pallets in pallet changing unit		Select change or no change of the pallet and specify the maximum number of pallets. 0 or 1: Not to change pallet 2 to 255: To change pallet (Numeric value indicates the maximum number of pallets.)
	Program type	M	
	Conditions	At power on	**Note:** When this parameter is set to 0 or 1, use of the pallet changing unit is prohibited.
	Unit	—	
	Setting range	0 to 255	

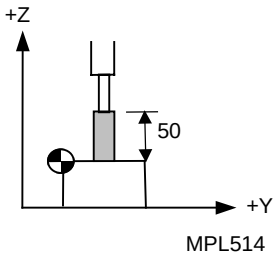
Classification	MACHINE	Display title	TABLE

Address	Meaning		Description
L47	To prepare or not to prepare next pallet change		Select preparation of next pallet or not to do so according to the pallet change mechanism. 0: Not to prepare next pallet 1: To prepare next pallet **Note:** When this parameter is set to 1, it is possible to set the number of the next pallet in the pallet changing unit.
	Program type	M	
	Conditions	At power on	
	Unit	—	
	Setting range	0, 1	
L48	Number of long boring bars		Specify the number of long boring bars mounted.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 9	
L49	Simultaneous operation of pallet change with ATC		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. 0: To operate ATC after pallet change 1: To operate pallet change and ATC simultaneously
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
L50	Rewriting of head number		Rewriting of head number in MDI mode: 0: Impossible 1: Possible (For five surface machining)
	Program type	—	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	

Classification	MACHINE	Display title	TABLE

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

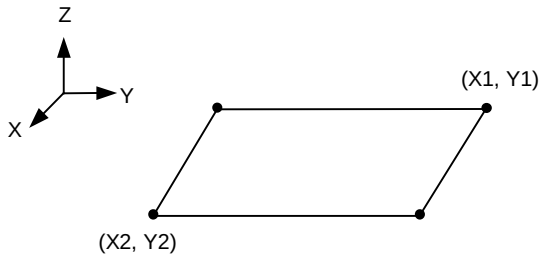
Classification	MACHINE	Display title	TABLE

Address	Meaning		Description
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) **Example:**  MPL514 - Mothod by M2 (distance from the tool tip to the zero point with a sign) .....–50 - Method by M32 (distance from the zero point to the tool tip with a sign) 50
	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 an EIA/ISO program. 0: Impossible 1: Possible
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
L58	Head index angle indication system		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)
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	

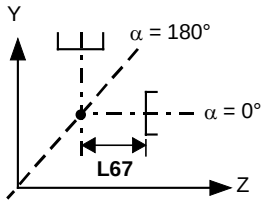
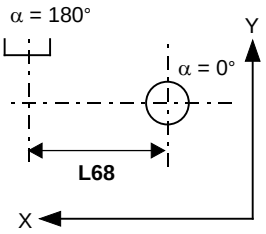
Classification	MACHINE	Display title	TABLE

Address	Meaning		Description
L59	Input selection for **HEAD OFFSET** display		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)
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
L60	Head quantity		The total number of heads to be mounted in the spindle (For five surface machining)
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 10	
L61	Output timing of AHC and APC		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)
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 2	

Classification	MACHINE	Display title	TABLE

Address	Meaning		Description
L62	Head relay point X1		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 sytem). For AGX machines, 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.  MPL515 (For five-surface machining) (For AGX series)
	Program type	M	
	Conditions	Immediate	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to ±99999999	
L63	Head relay point Y1		
	Program type	M	
	Conditions	Immediate	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to ±99999999	
L64	Head relay point X2		
	Program type	M	
	Conditions	Immediate	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to ±99999999	
L65	Head relay point Y2		
	Program type	M	
	Conditions	Immediate	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to ±99999999	

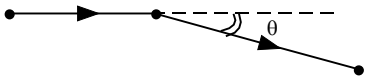
Classification	MACHINE	Display title	TABLE

Address	Meaning		Description
L66	Return/No return to head indexing point Z		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)
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
L67	Length between the end surface of the spindle and the center of head rotation		Set the length from the end surface of the spindle to the center of head rotation for respective machines. (100 mm in usual)  MPL516
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to ±99999999	
L68	Head correction value X		Set for respective machines.  MPL517 (For AGX series)
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to ±99999999	
L69	Head correction value Y		Set for respective machines. (For AGX series)
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to ±99999999	

Classification	MACHINE	Display title	TABLE

Address	Meaning		Description
L70	Axis movement from machining face on escapement		Specify the axes that simultaneously move from a machining face to the next machining face or in case of tool replacement. 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. 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. (For AGX series)
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	1, 0	
L71	Shift of basic coordinate for oblique face machining		For execution of the program for oblique face machining, specify to execute or not machining on the coordinate that is turned from the basic corrdinate (set in **WPC** unit or in **OFFSET** unit) at an angle of the correction value for the B-axis. 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 (For AGX series)
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	1, 0	
L72	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
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	msec	
	Setting range	0 to 56	

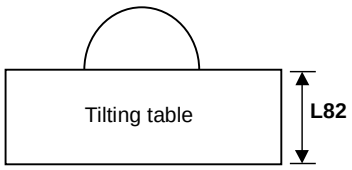
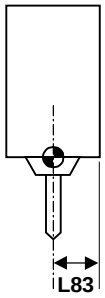
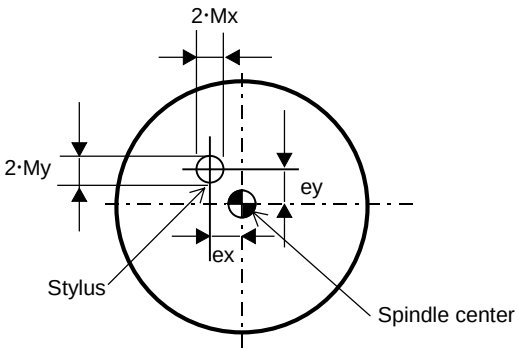
Classification	MACHINE	Display title	TABLE

Address	Meaning		Description
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	—	
	Unit	mm/min	
	Setting range	1 to 999999	
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	—	
	Unit	msec	
	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	—	
	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	—	
	Unit	1°	
	Setting range	0 to 30	

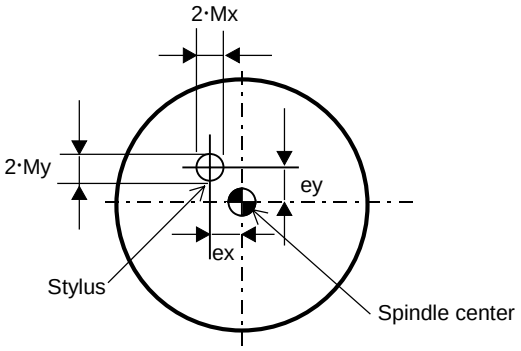
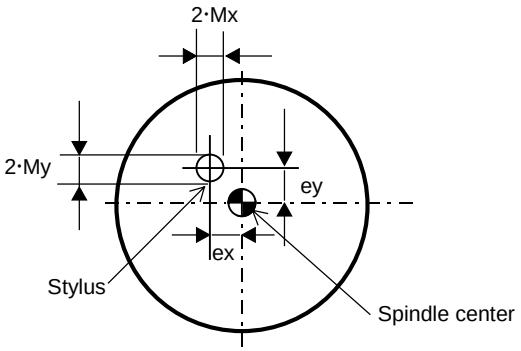
Classification	MACHINE	Display title	TABLE

Address	Meaning		Description
L78	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
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	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
L81	Fixed value (0)		
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

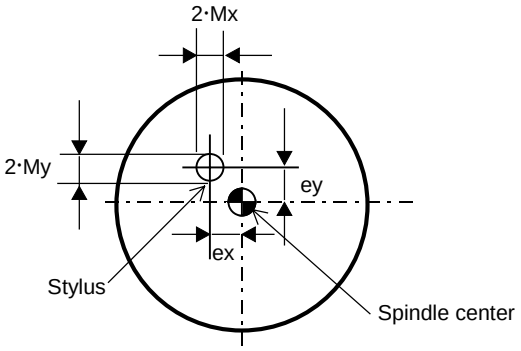
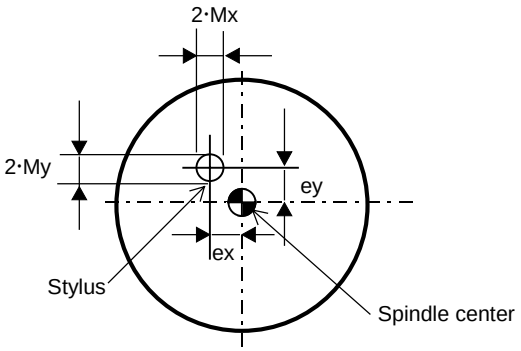
Classification	MACHINE	Display title	TABLE

Address	Meaning		Description
L82	Table thickness		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°).  (For machines equipped with a tilting table)
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.0001 mm	
	Setting range	±99999999	
L83	Spindle head radius		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.  (For machines equipped with a tilting table)
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.0001 mm	
	Setting range	±99999999	
L84	Correction value of alignment deviation X (Upper face)		 MPL519 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**) **Note:** The data is set automatically by execution of calibration measurement (on the upper face) with the **MMS** unit. (For five-surface machining)
	Program type	M	
	Conditions	After stop of movement	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to ±99999999	
L85	Correction value of alignment deviation Y (Upper face)		
	Program type	M	
	Conditions	After stop of movement	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to ±99999999	

Classification	MACHINE	Display title	TABLE

Address	Meaning		Description
L86	Correction value of alignment deviation X (0-degree face)		 MPL519 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**) **Note:** The data is set automatically by execution of calibration measurement (0-degree face) with the **MMS** unit. (For five-surface machining)
	Program type	M	
	Conditions	After stop of movement	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to ±99999999	
L87	Correction value of alignment deviation Y (0-degree face)		
	Program type	M	
	Conditions	After stop of movement	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to ±99999999	
L88	Correction value of alignment deviation X (90-degree face)		 MPL519 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**) **Note:** The data is set automatically by execution of calibration measurement (90-degree face) with the **MMS** unit. (For INTEGREX series) (For five-surface machining)
	Program type	M	
	Conditions	After stop of movement	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to ±99999999	
L89	Correction value of alignment deviation Y (90-degree face)		
	Program type	M	
	Conditions	After stop of movement	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to ±99999999	

Classification	MACHINE	Display title	TABLE

Address	Meaning		Description
L90	Correction value of alignment deviation X (180-degree face)		 MPL519 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**) **Note:** The data is set automatically by execution of calibration measurement (180-degree face) with the **MMS** unit. (For INTEGREX series) (For five-surface machining)
	Program type	M	
	Conditions	After stop of movement	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to ±99999999	
L91	Correction value of alignment deviation Y (180-degree face)		
	Program type	M	
	Conditions	After stop of movement	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to ±99999999	
L92	Correction value of alignment deviation X (270-degree face)		 MPL519 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**) **Note:** The data is set automatically by execution of calibration measurement (270-degree face) with the **MMS** unit. (For five-surface machining)
	Program type	M	
	Conditions	After stop of movement	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to ±99999999	
L93	Correction value of alignment deviation Y (270-degree face)		
	Program type	M	
	Conditions	After stop of movement	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to ±99999999	

Classification	MACHINE	Display title	TABLE

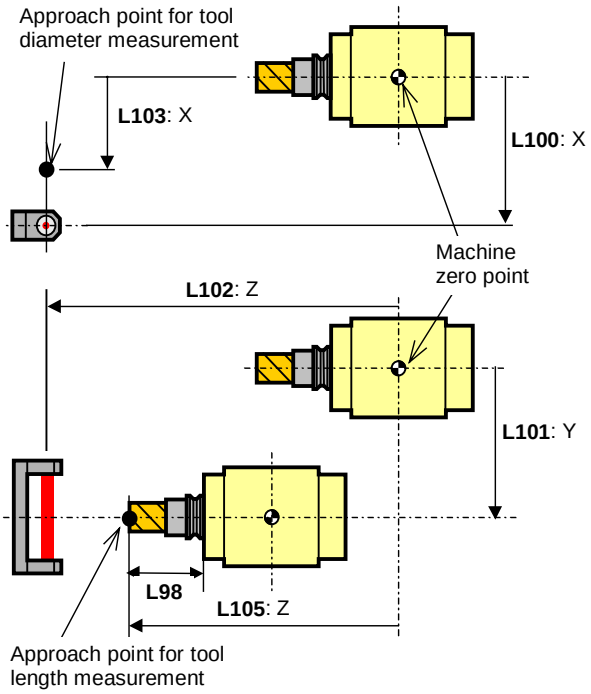
Address	Meaning		Description
L94	X/Y travel distance during EIA-programmed tool measurement		The X/Y travel distance can be set during the selection of EIA-programmed fully automatic tool length measurement on the **TOOL OFFSET** display. 0: Invalid 1: Valid **Note:** Valid only when the measuring equipment to be used for the tool measurement is a measuring table (**L106** bit 0 = 0).
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
L95	Offset number auto-setting for EIA-programmed tool measurement		The offset number is auto-set during EIA-programmed tool measurement data setting. 0: Invalid 1: Valid **Note:** When making the auto-setting function valid, see the description of **L96**.
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
L96	Offset for EIA-programmed tool measurement		The amount of shifting for TNo. during offset number auto-setting for EIA-programmed tool measurement data setting. [Offset No.] = [TNo. setting] + [**L96** setting] **Note:** Valid only when **L95** = 1.
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 4000	
L97	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

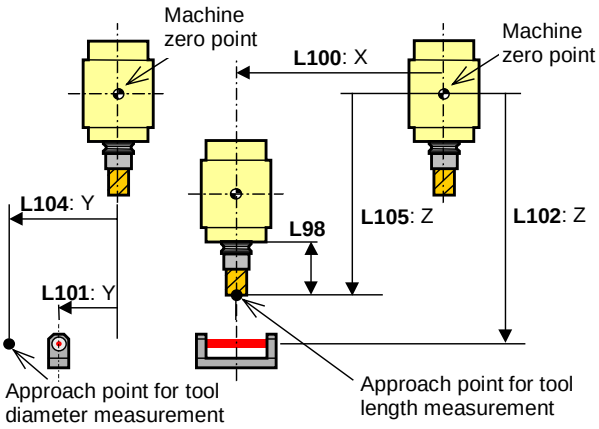
Classification	MACHINE

Display title	TABLE

Address	Meaning		Description
L98	Max. tool length for laser tool length measurement		Specify the muximum tool length for the MDI laser tool length measurement.
	Program type	M, E	
	Conditions	At power on	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	99999999	
L99	Cycle time for saving the operational status management data		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. **Notes:** **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

<table>
<tr><th>Address</th><th>Meaning</th><th>Description</th></tr>
<tr><td>L100
to
L105</td><td>
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
<table>
<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 inch</td></tr>
<tr><td>Setting range</td><td>±99999999</td></tr>
</table>
</td><td>
[INTE e-HII]


[INTE e-VII]


</td></tr>
<tr><td>L106
(bit 0)</td><td>
Measuring equipment selection
<table>
<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>
</td><td>
0: Measuring table
1: Laser
Note:
Set this parameter to 0, if TOOL EYE is used.
</td></tr>
</table>

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)	Selection of whether workpiece measurement and tool measurement results are to be stored into tool data of the lower turret		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

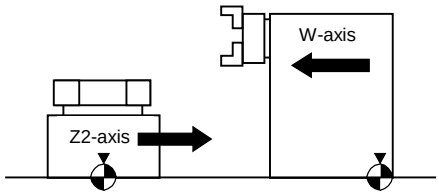
Address	Meaning		Description
L106 (bit 5)	Selection of measurement execution timing		Select the timing to execute the workpiece and tool measurement. 0: The measurement unit is executed whenever the parts count reaches same or a multiple of the specified measurement interval in addition to the first time. 1: 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 6)	Selection of tool measurement operation		Select the tool measurement operation. 0: Offset judgement - wear offset invalid 1: Offset judgement - wear offset valid
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
L107 (bit 0)	Tool path drawing		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. 0: Loft on the standard coordinates system 1: Loft on the machine coordinates system **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. (For AGX series)
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
L107 (bit 1)	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

Classification	MACHINE

Display title	TABLE

Address	Meaning		Description
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
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
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
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
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. (For AGX series)
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	

Classification	MACHINE	Display title	TABLE

Address	Meaning		Description
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. (For AGX series)
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
L108	Fixed value (0)		
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
L109 (bit 0) to (bit 3)	Axial direction for checking for interference at software limit 4 Bit 0: Interference axial direction (1st set) Bit 1: Interference axial direction (2nd set) Bit 2: Interference axial direction (3rd set) Bit 3: Interference axial direction (4th set)		Specify the relationship between axial directions of the two axes to be used to check for interference. 0: Same direction 1: Reverse directions Even if the axes for the interference check are present on the same sliding surface, the axial directions of the two axes may differ as shown below. In this case set 1. 
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	—	
	Setting range	0, 1	
L110 (bit 0)	—		Opposed-spindle lathe specifications
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	

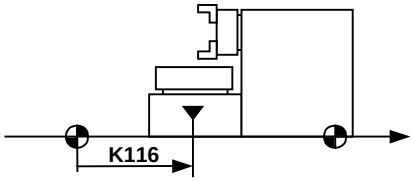
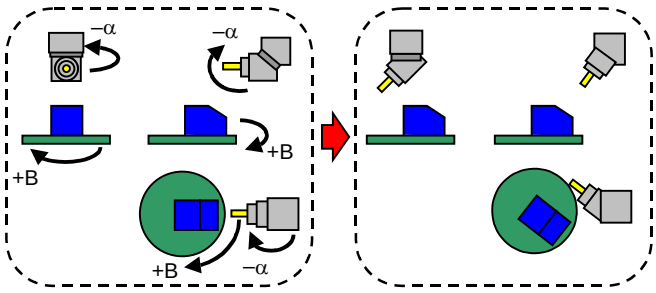
Classification	MACHINE	Display title	TABLE

Address	Meaning		Description
L110 (bit 1)	—		Vertically inverted spindle specifications
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
L110 (bit 2)	—		Axis name of the secondary spindle 0: Z/C-axis 1: W/U-axis
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
L110 (bit 3)	—		Specify whether to disable or enable the display of “section to be machined” in the milling tool sequence. 0 : Disabled 1 : Enabled
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
L110 (bit 4)	—		Display of the **BUFFER**, **REMAIN** and **POSITION** information during coordinate conversion. 0: Real axis display 1: Virtual axis display
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	

Classification	MACHINE	Display title	TABLE

Address	Meaning		Description
L110 (bit 7)	—		Z-axis direction 0: Horizontal 1: Vertical
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
L111 **L112**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
L113 to **L116**	Interference check reference axis **L113** 1st set of axes **L114** 2nd set of axes **L115** 3rd set of axes **L116** 4th set of axes		Set one of the axes to be used for the interference check. Specify the axis number by that of NC. The checking function will be invalid if the number is missing or if an invalid number is set.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 16	
L117 to **L120**	Interference check axis **L117** 1st set of axes **L118** 2nd set of axes **L119** 3rd set of axes **L120** 4th set of axes		Set the interference check target axis with respect to the interference check reference axis. Specify the axis number by that of NC. The checking function will be invalid if the number is missing or if an invalid number is set.
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	0	

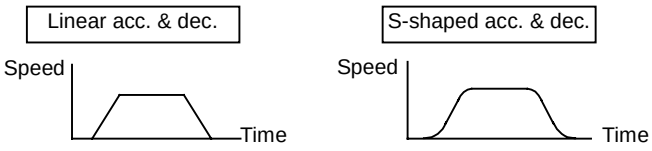
Classification	MACHINE	Display title	TABLE

Address	Meaning		Description
L121 to L124	Interference clearance **L121** 1st set of axes **L122** 2nd set of axes **L123** 3rd set of axes **L124** 4th set of axes		Set the machine coordinate of the interference check target axis that is about to cause interference when the interference check reference axis is at its zero point. In the example shown below, if the W-axis and the Z2-axis are defined as the interference check reference axis and the interference check target axis, respectively, set the machine coordinate of the Z2-axis existing at where the lower turret and the secondary spindle are most likely to interfere. If zero is entered, the checking function will be invalid. 
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	0	
L125	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
L126	Positioning direction of the head rotation α-axis for oblique plane indexing		Positioning direction of the α-axis for oblique plane indexing specified in **ANGLE** (plane angle) of the surface definition sequence. 0 : Makes the sign of the plane angle value invalid and positions the α-axis in its minus direction. 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. 2 : Makes the sign of the plane angle value invalid and positions the α-axis in its plus direction. If the setting of this parameter is other than the above, the value of **L126** will be handled as 1. <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°  (For AGX series)
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 2	

Classification	MACHINE	Display title	TABLE

Address	Meaning		Description
L127 **L128**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
L129 **L130**	Acceleration/deceleration filter (1st stage) **L129** G1 time constant **L130** G0 time constant		This parameter functions as a filter to smoothen the waveform command specified for pre-interpolation acceleration/deceleration. <Type of feed and acceleration/deceleration> L = Post-interpolation Linear acceleration/deceleration S = Pre-interpolation S-shaped acceleration/deceleration
	Program type	M, E	
	Conditions	After stop of axis movement	
	Unit	msec	
	Setting range	0 to 200	
L131 **L132**	Acceleration/deceleration filter (2nd stage) **L131** G1 time constant **L132** G0 time constant		
	Program type	M, E	
	Conditions	After stop of axis movement	
	Unit	msec	
	Setting range	0 to 200	
L133 to **L138**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

		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	



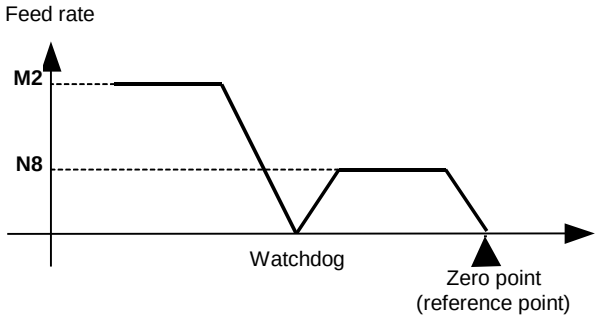
Classification	MACHINE

Display title	TABLE

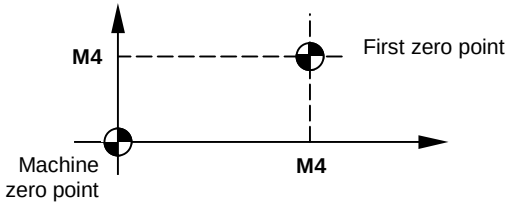
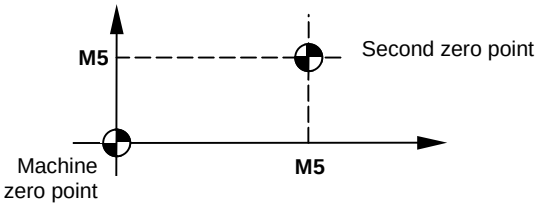
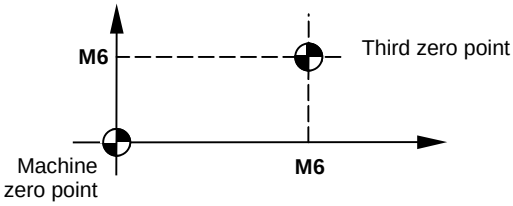
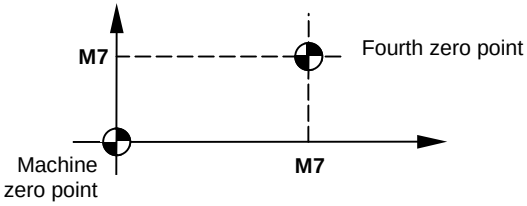
Address	Meaning		Description
L139	Minimum usable tool diameter of the measurable chamfering tool		Used for fully automatic tool measurement with a measuring table.
	Program type	—	
	Conditions	Immediate	
	Unit	0.1 mm/0.01 inch	
	Setting range	0 to 99.9 mm/9.99 inch	
L140 to **L144**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

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

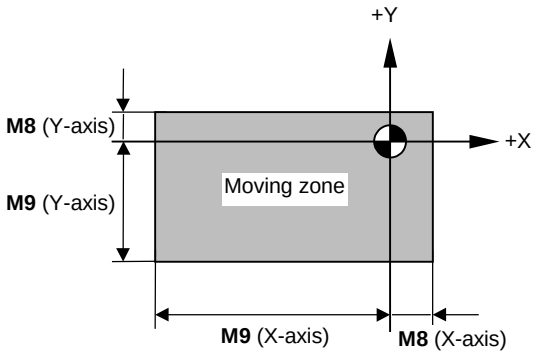
Classification	MACHINE	Display title	FEED VEL.

Address	Meaning		Description
M1	Rapid feed rate		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
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	1 mm/min (1°/min)	
	Setting range	0 to 600000	**Note:** Initial zero-point return is performed at the feed rate set in parameter **M2**.
M2	Feed rate for initial zero-point return		The feed rate for moving each axis during initial zero-point return (reference-point return) at power on  MPL520
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	1 mm/min (1°/min)	
	Setting range	0 to 600000	
M3	Cutting feed rate limit		The limit of cutting feed rate during automatic operation Even if a feed rate higher than this parameter setting is specified, the latter governs.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	1 mm/min (1°/min)	
	Setting range	0 to 600000	

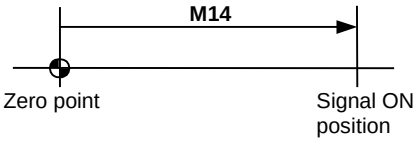
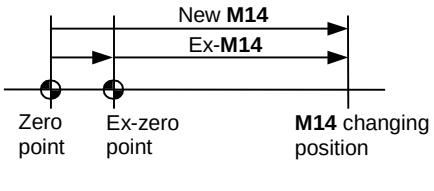
Classification	MACHINE	Display title	FEED VEL.

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

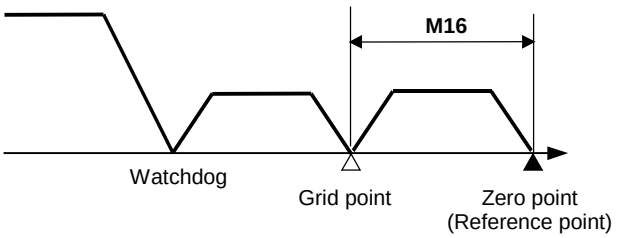
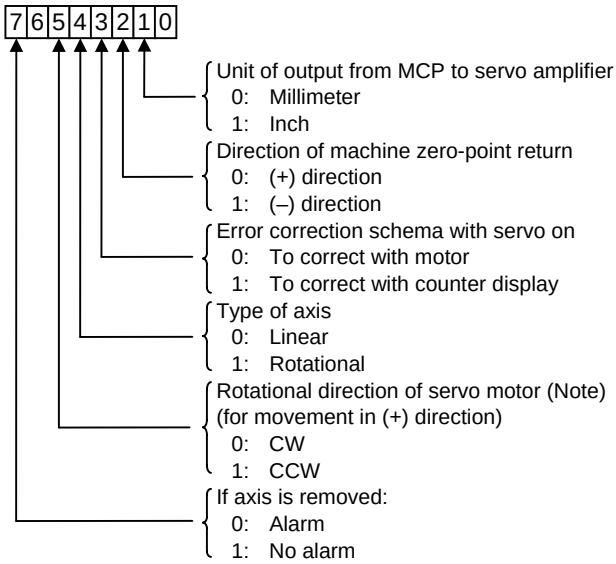
Classification	MACHINE	Display title	FEED VEL.

Address	Meaning		Description
M8	Maximum software limit specified by manufacturer (+ direction)		The maximum moving zone permissible under the machine specifications Set the machine coordinate values. **Example:**  MPL522
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	0.0001 mm/0.0001 deg	
	Setting range	±99999999	
M9	Maximum software limit specified by manufacturer (– direction)		**Note:** This parameter is invalid when **M8** = **M9**.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	0.0001 mm/0.0001 deg	
	Setting range	±99999999	
M10	Command unit		Select a unit of command from the following table and set it. * micron system 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. 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 Fixed value
	Program type	M, E	
	Conditions	At power on	
	Unit	—	
	Setting range	&0 to &7F	

Classification	MACHINE	Display title	FEED VEL.

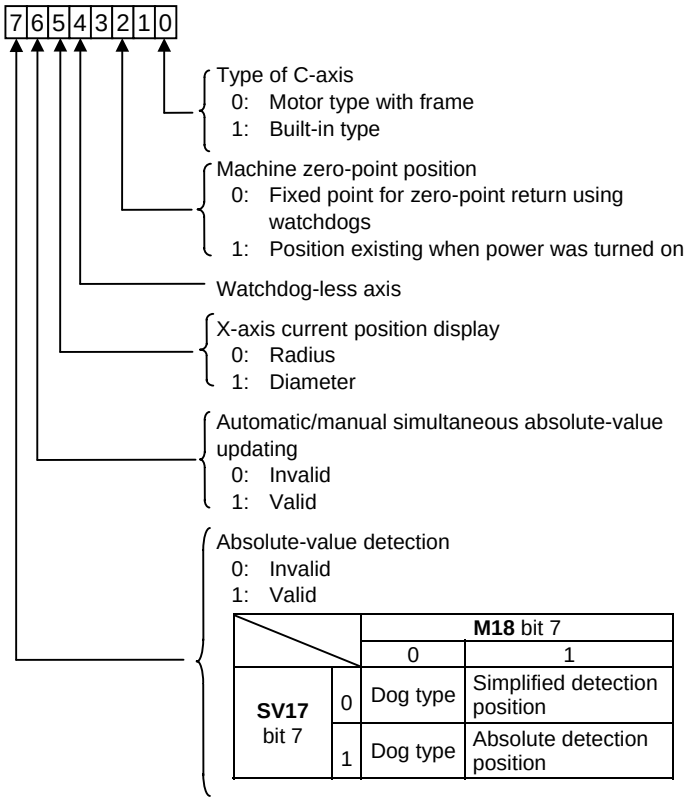
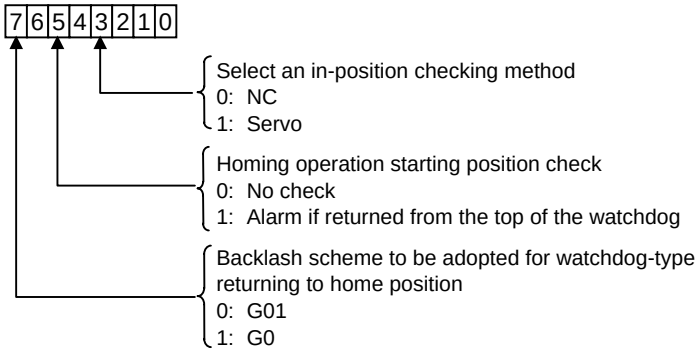
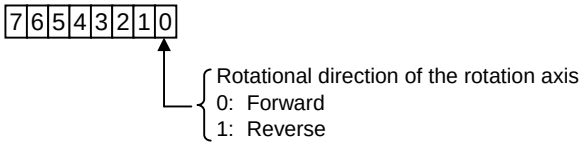
Address	Meaning		Description
M12	Coding of incremental axis		Register the incremental axes respectively in hexadecimal numbers of the ASCII code.
	Program type	M, E	
	Conditions	At power on	
	Unit	—	
	Setting range	&0 to &7F	
M13	Axis name (for display)		Specify the address name of the axes to be used for display, by the appropriate hexadecimal number of the ASCII code. For reverse display, set up the most significant bit. **Example 1:** Reverse display of “C” C : &0043 → Reverse display of “C”: &00C3 **Example 2:** Reverse display of “X1” X1 : &5831 → Reverse display of “X1”: &D831
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	&0 to &7F	
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 deg	
	Setting range	±99999999	

Classification	MACHINE	Display title	FEED VEL.

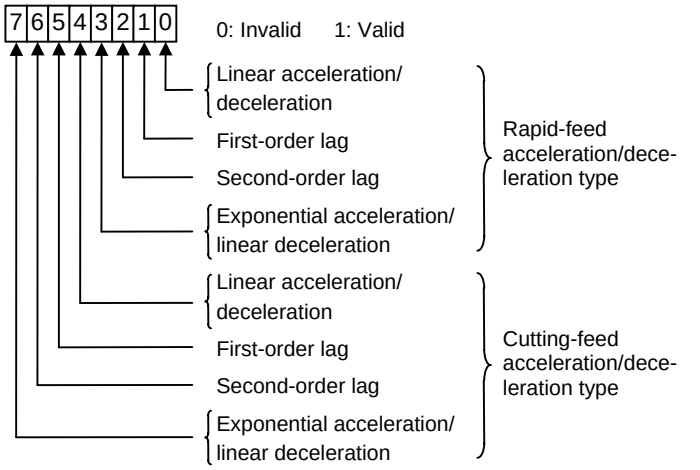
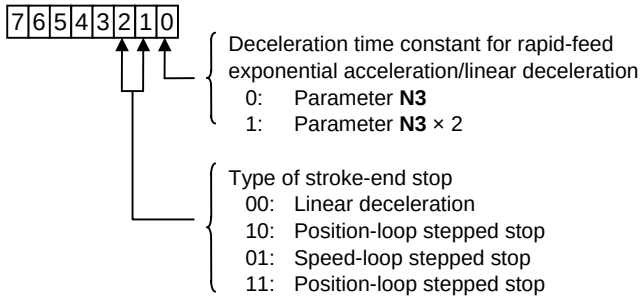
Address	Meaning		Description
M15	Axis name (for axis name changing)		Specify the name of the address to be used for axis name changing (G110), by the appropriate hexadecimal number of the ASCII code.
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	&0 to &7F	
M16	Zero-point shift amount		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.  MPL527
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.0001 mm/0.0001 deg	
	Setting range	0 to 99999999	
M17	Axis control flag		 **Note:** Usually, specify the rotational direction for the motor when viewed from the front (the opposite side of the encoder). For “Spindle-Cs control”, however, specify the rotational direction for the motor when viewed from the rear.
	Program type	M, E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	Binary, eight digits	

Classification	MACHINE

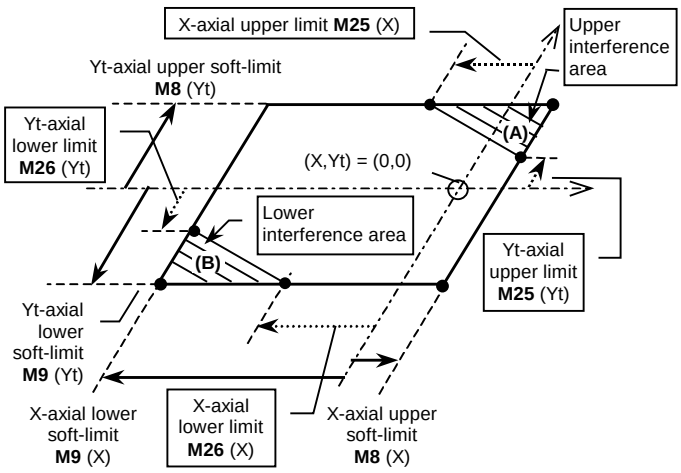
Display title	FEED VEL.

Address	Meaning		Description
M18	Axis control flag		 **M18** bit 7 / **SV17** bit 7: 0 → Dog type / Simplified detection position; 1 → Dog type / Absolute detection position
	Program type	M, E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	Binary, eight digits	
M19	Axis control flag		
	Program type	M, E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	Binary, eight digits	
M20	Axis control flag		
	Program type	M, E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	Binary, eight digits	

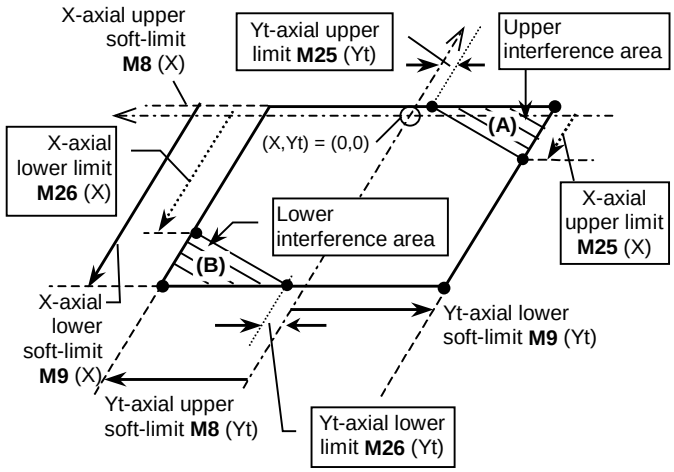
Classification	MACHINE	Display title	FEED VEL.

Address	Meaning		Description
M21	—		 **Note:** Time constants for each type of acceleration/deceleration control must be set using parameters **N1** through **N6**.
	Program type	M, E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	Binary, eight digits	
M22	—		
	Program type	M, E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	Binary, eight digits	
M23 **M24**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

Classification	MACHINE	Display title	FEED VEL.

Address	Meaning		Description
M25 (Type A)	Illegal axis area upper limits (Type A)		Set an access inhibition area in the inclined-axis control software limit area consisting of the X-axis and the Yt-axis. - 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: 1. **M26** (B-axis) ≤ B-axis angle ≤ **M25** (B-axis) 2. **M25** (B-axis) = **M26** (B-axis) = 0 <Type A (**BA126** bit 1 = 0)>  Upper: Shaded area (A) in the above diagram, determined by parameters **M8** (X), **M8** (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. Lower: Shaded area (B) in the above diagram, determined by parameters **M9** (X), **M9** (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.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	0.0001 mm/0.0001 deg	
	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 deg	
	Setting range	±99999999	

Classification	MACHINE	Display title	FEED VEL.

Address	Meaning		Description
M25 (Type B)	Illegal axis area upper limits (Type B)		Set an access inhibition area in the inclined-axis control software limit area consisting of the X-axis and the Yt-axis. - 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: 1. **M16** (B-axis) ≤ B-axis angle ≤ **M25** (B-axis) 2. **M25** (B-axis) = **M26** (B-axis) = 0 <Type B (**BA126** bit 1 = 1)>  Upper: 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. Lower: 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.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	0.0001 mm/0.0001 deg	
	Setting range	±99999999	
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 deg	
	Setting range	±99999999	

Classification	MACHINE	Display title	FEED VEL.

Address	Meaning		Description
M27	Optimum acceleration control (Target speed)		Set the value that determines the maximum allowable value (Amax) of the acceleration that occurs between blocks. $Amax = \frac{\mathbf{M27}}{\mathbf{N25}} \times 60 \times 1000 \times \frac{100 - \mathbf{N26}}{100}$ (mm/msec^2) **Note:** 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)
	Program type	M, E	
	Conditions	After stop of axis movement	
	Unit	mm/min	
	Setting range	1 to 5000	
M28	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
M29	Rapid feed clamping speed 1 for superposition control		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.) - 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: **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 **Note:** Whether the axes move in the same direction or in different directions is judged from the command and polarity (parameter **N21**, bit 6).
	Program type	M, E	
	Conditions	After stop of axis movement	
	Unit	1 mm/min	
	Setting range	0 to 600000	
M30	Rapid feed clamping speed 2 for superposition control		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.) - Set **M29** to **M31** so that the following conditions are satisfied: **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 **Note:** Whether the axes move in the same direction or in different directions is judged from the command and polarity (parameter **N21**, bit 6).
	Program type	M, E	
	Conditions	After stop of axis movement	
	Unit	1 mm/min	
	Setting range	0 to 600000	

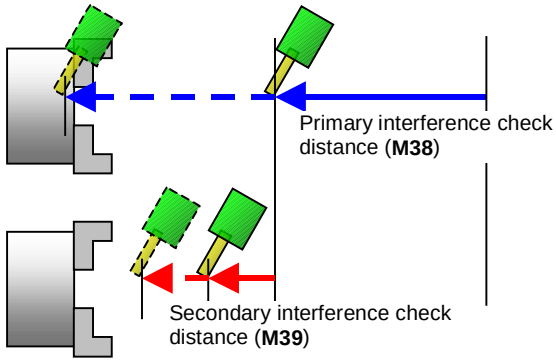
Classification	MACHINE	Display title	FEED VEL.

Address	Meaning		Description
M31	Cutting feed clamping speed for superposition control		Set the cutting feed clamping speed for superposition control. (Set value for the reference axis and superposition axis to be used during superposition control.) - Set **M29** to **M31** so that the following conditions are satisfied: **M29** (RA) + **M31** (SA) ≤ **M1** (SA) **M31** (RA) + **M29** (SA) ≤ **M1** (SA) **M30** (RA) + **M30** (SA) ≤ **M1** (SA) **M31** (RA) + **M31** (SA) ≤ **M1** (SA) RA: reference axis SA: superposition axis
	Program type	M, E	
	Conditions	After stop of axis movement	
	Unit	1 mm/min	
	Setting range	0 to 600000	
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	1 mm/min / 1 deg/min	
	Setting range	0 to 99999999	
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	1 mm/min / 1 deg/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	1 mm/min / 1 deg/min	
	Setting range	0 to 99999999	

Classification	MACHINE	Display title	FEED VEL.

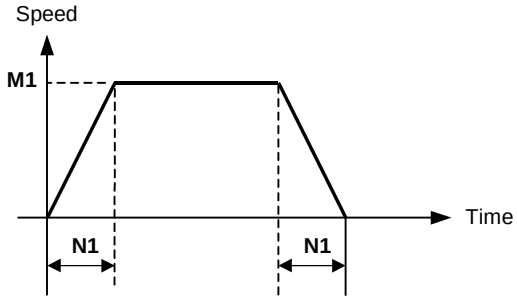
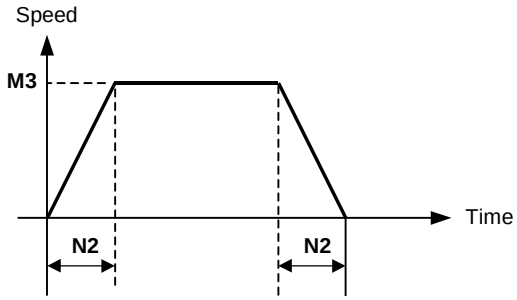
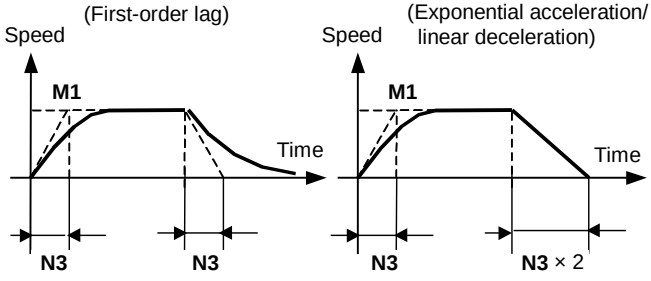
Address	Meaning		Description
M35	Safety clamping speed (Safety supervisory mode 3)		Set the safety clamping speed to be used in safety supervisory mode 3. In safety supervisory mode 3 and while a safety clamping request is in effect, operation decelerates to the set speed. (Assign a value smaller than that of parameter **M33** to parameter **M35**.)
	Program type	M, E	
	Conditions	After stop of axis movement	
	Unit	1 mm/min / 1 deg/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	
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	

Classification	MACHINE	Display title	FEED VEL.

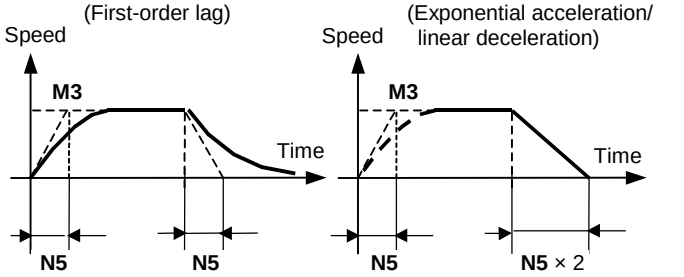
Address	Meaning		Description
M38 **M39**	Interference check distances for Intelligent Safety Shield **M38**: Primary check distance **M39**: Secondary check distance		Set the machine interference detection distances for the Intelligent Safety Shield. Set data so that the primary check distance is longer than the secondary check distance.  **Note:** When inch is set for units of data setting (**K105** bit 7 =1), inch system is also used to set this parameter.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	0.0001 mm/0.00001 inch/ 0.0001 deg	
	Setting range	0 to 99999999	
M40 to **M48**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

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

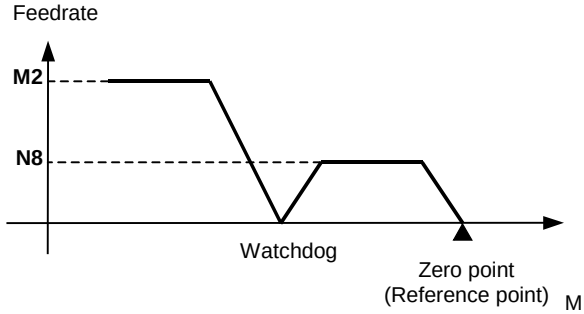
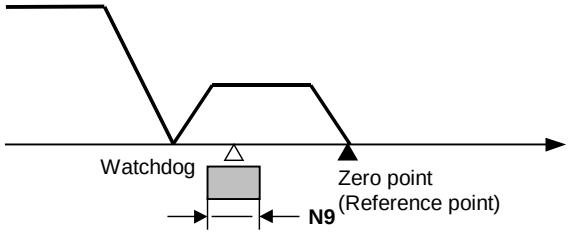
Classification	MACHINE	Display title	TIME CONST.

Address	Meaning		Description
N1	Rapid-feed time constant (linear acceleration/deceleration)		Set the time constant to obtain linear acceleration/deceleration of the rapid feed rate.  MPL523 **Note:** This parameter is valid only when bit 0 of **M21** is 1.
	Program type	M, E	
	Conditions	Immediate	
	Unit	1 msec	
	Setting range	4 to 1800	
N2	Cutting-feed time constant (linear acceleration/deceleration)		Set the time constant to obtain linear acceleration/deceleration of the cutting feed rate.  MPL523 **Note:** This parameter is valid only when bit 4 of **M21** is 1.
	Program type	M, E	
	Conditions	Immediate	
	Unit	1 msec	
	Setting range	4 to 1800	
N3	Rapid-feed time constant (First-order lag)		First-order lag time constant for rapid-feed acceleration/deceleration  MPL523 **Note:** This parameter is valid only when either bit 1, 2 or 3 of **M21** is 1.
	Program type	M, E	
	Conditions	Immediate	
	Unit	1 msec	
	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 smoothens 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	1 msec	
	Setting range	0 to 455	
N5	Cutting-feed time constant (First-order lag)		First-order lag time constant for cutting-feed acceleration/ deceleration  MPL524 **Note:** This parameter is valid only when either bit 5, 6 or 7 of **M21** is 1.
	Program type	M, E	
	Conditions	Immediate	
	Unit	1 msec	
	Setting range	0 to 5000	
N6	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
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	1 msec	
	Setting range	1 to 32767	

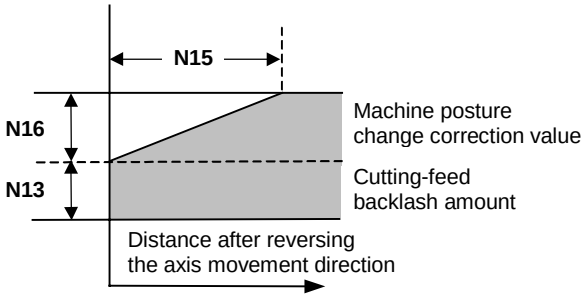
Classification	MACHINE	Display title	TIME CONST.

Address	Meaning		Description
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.  MPL525
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	1 mm/min (1°/min)	
	Setting range	1 to 60000	
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 intial 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.  MPL526 **Note:** The amount of grid ignorance must not exceed 1 grid spacing.
	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, as is the case, with a linear scale.
	Program type	M, E	
	Conditions	At power on	
	Unit	mm (0.001°)	
	Setting range	0 to 32767	
N11	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

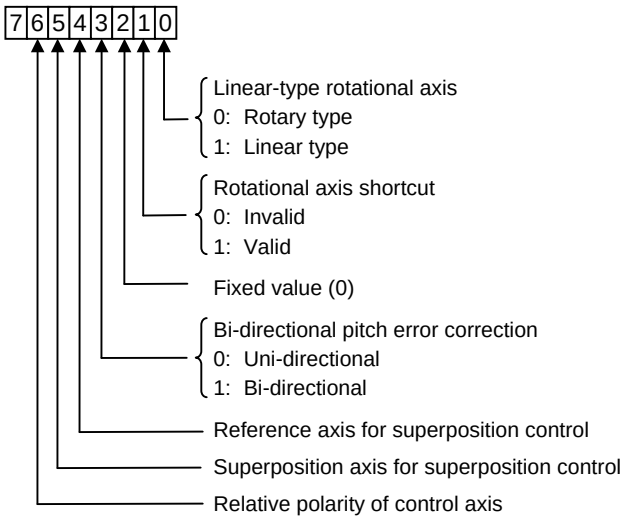
Classification	MACHINE	Display title	TIME CONST.

Address	Meaning		Description
N12	Rapid-feed backlash		The backlash amount to be corrected after the axis movement direction has been reversed in either the rapid-feed (G00) mode or manual mode (except handle-pulse feed mode) **Note:** Setting conditions: **N12** < **N13**
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.00005 mm/0.00005 deg	
	Setting range	±9999	
N13	Cutting-feed backlash		The backlash amount to be corrected after the axis movement direction has been reversed in either the cutting-feed (G01) mode or manual handle-pulse feed mode. **Note:** Setting conditions: **N12** < **N13**
	Program type	M, E	
	Conditions	At power on	
	Unit	0.00005 mm/0.00005 deg	
	Setting range	±9999	
N14	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

Classification	MACHINE	Display title	TIME CONST.

Address	Meaning		Description
N15	Width to which the machine posture change correction is to be applied		
	Program type	M, E	
	Conditions	At power on	
	Unit	0.00005 mm/0.00005 deg	
	Setting range	0 to 32767	
N16	Machine posture change correction value		
	Program type	M, E	
	Conditions	At power on	
	Unit	0.00005 mm/0.00005 deg	
	Setting range	0 to 65535	
N17	Servo amplifier channel number		Specify the channel number of the servo amplifier. 1: 1 ch 2: 2 ch 3: 3 ch
	Program type	M, E	
	Conditions	At power on	
	Unit	—	
	Setting range	0 to 3	
N18	Servo amplifier rotary switch number		Specify the rotary switch number of the servo 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	

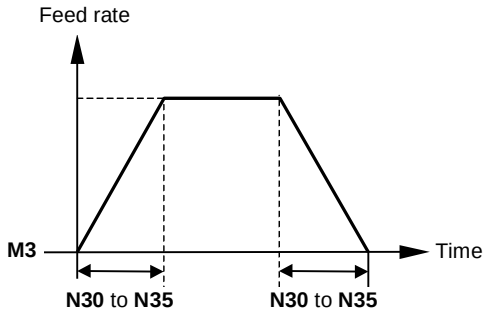
Classification	MACHINE	Display title	TIME CONST.

Address	Meaning		Description
N19	Axis system number		Set the system number of the axes. 0: System 1 1: System 2 2: System 3 3: System 4
	Program type	M, E	
	Conditions	At power on	
	Unit	—	
	Setting range	0 to 3	
N20	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
N21	—		
	Program type	M, E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	Binary, eight digits	

Classification	MACHINE	Display title	TIME CONST.

Address	Meaning		Description
N22 to **N24**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
N25	Time constant for deceleration rate calculation		Set the maximum allowable value, Amax, for the acceleration occurring between blocks. $Amax = \frac{\mathbf{M27}}{\mathbf{N25}} \times 60 \times 1000 \times \frac{100 - \mathbf{N26}}{100}$ (mm/msec^2) **Note:** 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)
	Program type	M, E	
	Conditions	After stop of axis movement	
	Unit	mm/min	
	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		Set the rapid feed (linear acceleration/deceleration) time constant for superposition control. **Note:** Set the same value for all axes.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	1 msec	
	Setting range	4 to 1800	

Classification	MACHINE	Display title	TIME CONST.

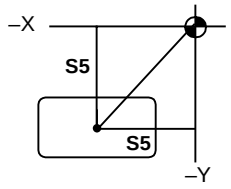
Address	Meaning		Description
N28	Cutting feed time constant for superposition		Set the cutting feed (linear acceleration/deceleration) time constant for superposition control. **Note:** Set the same value for all axes.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	1 msec	
	Setting range	4 to 1800	
N29	Time constant for shape correction rapid feed acceleration/deceleration filter		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.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	msec	
	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		Set the cutting feed time constant for a time constant changeover M-code command. 
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	msec	
	Setting range	0 to 1800	
N36 to **N48**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

2-3-14 Machine parameter ANOTHER (S)

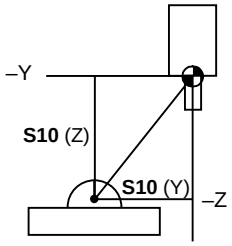
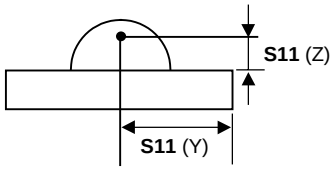
Classification	MACHINE	Display title	ANOTHER

Address	Meaning		Description
S1 **S2**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
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	
S4	Feed forward gain		Set for each axis the feed forward gain for acceleration/deceleration before interpolation.
	Program type	M, E	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 100	

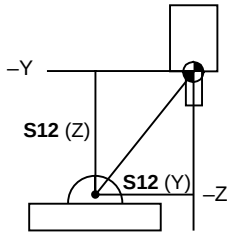
Classification	MACHINE	Display title	ANOTHER

Address	Meaning		Description
S5	Rotational center of the table		Set the position of the rotational center of the table in the machine coordinates system.  **Notes:** **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) (For dynamic offset) (For HV machining) (For machines equipped with a tilting table)
	Conditions	At power on	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to ±99999999	
S6	Absolute position detection parameter		When movement is beyond the length set by this parameter during the power off, it activates the alarm mode.
	Program type	M, E	
	Conditions	—	
	Unit	0.0001 mm/0.00001 inch/ 0.0001 deg	
	Setting range	0 to ±99999999	
S7	Upper limit (on Z-axis) of machining range for table rotating maching I		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.
	Program type	M, E	
	Conditions	At power on	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	0 to ±99999999	(For HV machining)

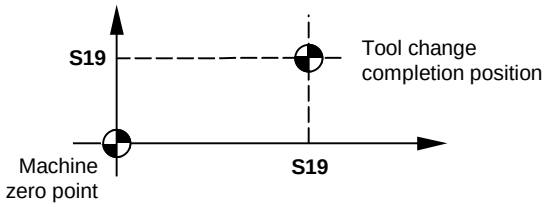
Classification	MACHINE	Display title	ANOTHER

Address	Meaning		Description
S8	Feed-forward gain G00		Specify the pre-interpolation acceleration/deceleration feed-forward gain levels for each axis.
	Program type	M, E	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 100	
S9	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
S10	Axis of rotation of the tilting table		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)
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.0001 mm	
	Setting range	0 to ±99999999	
S11	Corner position of the tilting table		Specify the corner position of the tilting table in (Y- and Z-axial) distances from its axis of rotation. This parameter is used for the software travel limit function. **Note:** X-axial data are not required.  (For machines equipped with a tilting table)
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.0001 mm	
	Setting range	0 to 99999999	

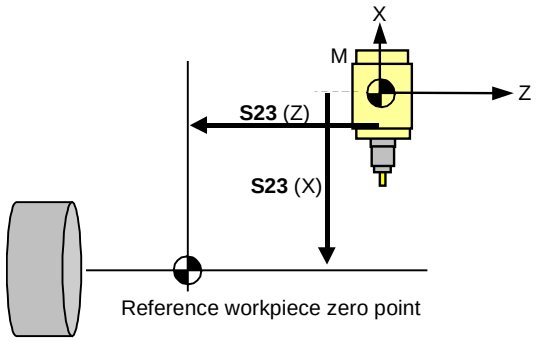
Classification	MACHINE	Display title	ANOTHER

Address	Meaning		Description
S12	Axis of rotation of the tilting table (Used for the automatic program origin calculation function)		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). Measure and enter data for respective machines. **Note:** X-axial data are not required.  (For machines equipped with a tilting table)
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to ±99999999	
S13	G00 in-position width		Set the in-position width for G00. The in-position check for G00 is effective when the parameter **K103** bit 7 is 1. For utilizing the in-position width of G00, set the in-position width of the servo parameter **SV024** to 0 to avoid trouble.
	Program type	M, E	
	Conditions	At power on	
	Unit	0.001 mm	
	Setting range	0 to 32767	
S14	G01 in-position width		Set the inposition 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 **K103** bit 7 set to 1. For utilizing the in-position width for G01, set the inposition width of the servo parameter **SV024** to 0 to avoid trouble.
	Program type	M, E	
	Conditions	At power on	
	Unit	0.001 mm	
	Setting range	0 to 32767	
S15	Amount of reference position correction (Only for bidirectional pitch error correction)		Specify by an absolute value the amount of reference position correction to be performed during the valid status of bidirectional pitch error correction when the axis is moved to a reference position from the direction opposite to that of a return to home position.
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.00005 mm	
	Setting range	–32768 to 32767	

Classification	MACHINE	Display title	ANOTHER

Address	Meaning		Description
S16	Unbalanced axis torque offset		Set automatically after estimation of the characteristics.
	Program type	M	
	Conditions	At power on	
	Unit	—	
	Setting range	—	
S17	Torque limit buffer reduction ratio 1		Specify torque limit buffer reduction ratio 1.
	Program type	M, E	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 200	
S18	Torque limit buffer reduction ratio 2		Specify torque limit buffer reduction ratio 2.
	Program type	M, E	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 200	
S19	Tool change completion position of the long boring bar end tool		Specify the tool change completion position of the long boring bar end tool by the corresponding machine coordinates.  MPL521
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.0001 mm/0.0001 deg	
	Setting range	±99999999	

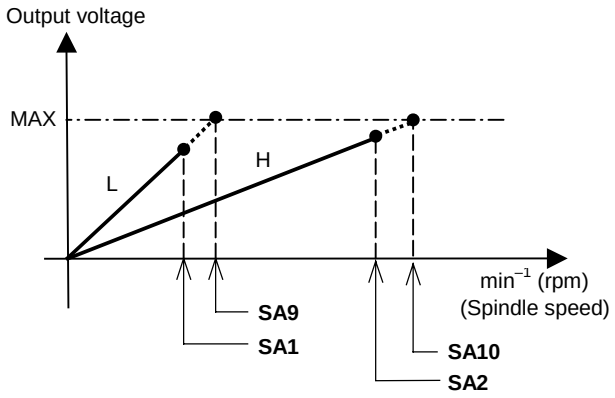
Classification	MACHINE	Display title	ANOTHER

Address	Meaning		Description
S20 **S21**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
S22	Cutting feed clamping speed during tool tip point control		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.
	Program type	E	
	Conditions	After stop of movement	
	Unit	1 mm/min (1°/min)	
	Setting range	1 to 200000	
S23	Reference workpiece zero point		Set the position of a reference workpiece zero point for each axis in the machine coordinate system. 
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	±99999999	
S24 to **S48**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

2-3-15 Machine parameter SPINDLE (SA)

Classification	MACHINE

Display title	SPINDLE

<table>
<tr><th>Address</th><th colspan="2">Meaning</th><th>Description</th></tr>
<tr><td rowspan="5">SA1
to
SA8</td><td colspan="2" rowspan="2">Maximum RPM of spindle in each speed range (range 1 to 8)</td><td>Number of revolutions per minute of the spindle in each speed range
<table><tr><td rowspan="2">Address</td><td colspan="4">Maximum number of speed ranges</td></tr><tr><td>1</td><td>2</td><td>3</td><td>4</td></tr><tr><td>SA1</td><td>○</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></table>
Example:
 
</td></tr>
<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>1 min−1 (rpm)</td></tr>
<tr><td>Setting range</td><td>0 to 99999</td></tr>
<tr><td rowspan="5">SA9
to
SA16</td><td colspan="2" rowspan="2">Constants for calculating each gear speed of the spindle (range 1 to 8)</td><td>Constants for calculating each gear speed of the spindle
<table><tr><td rowspan="2">Address</td><td colspan="4">Maximum number of speed ranges</td></tr><tr><td>1</td><td>2</td><td>3</td><td>4</td></tr><tr><td>SA9</td><td>○</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></table>
(⇨ SA1, SA2, SA3, SA4)</td></tr>
<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>1 min−1 (rpm)</td></tr>
<tr><td>Setting range</td><td>0 to 99999</td></tr>
</table>

Classification	MACHINE	Display title	SPINDLE

Address	Meaning		Description
SA17 to **SA24**	Maximum RPM of spindle during tapping cycle (range 1 to 8)		The maximum number of revolutions per minute of the spindle in each speed range during a tapping cycle
	Program type	M, E	
	Conditions	At power on	
	Unit	1 min^{-1} (rpm)	
	Setting range	0 to 99999	
SA25 to **SA32**	Spindle speed during gear shifting (range 1 to 8)		The number of revolutions per minute of the spindle during shifting of gears thru the various ranges
	Program type	M, E	
	Conditions	At power on	
	Unit	1 min^{-1} (rpm)	
	Setting range	0 to 32767	
SA33 to **SA40**	Acceleration/deceleration time constant for the synchronous tapping (range 1 to 8)		Linear acceleration/deceleration time constant for the synchronous tapping cycle **SA33:** Range 1 **SA34:** Range 2 **SA35:** Range 3 **SA36:** Range 4
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	1 min^{-1} (rpm)	
	Setting range	0 to 1800	
SA41	Spindle orientating speed		Specify the spindle orientating speed.
	Program type	M, E	
	Conditions	At power on	
	Unit	1 min^{-1} (rpm)	
	Setting range	0 to 32767	

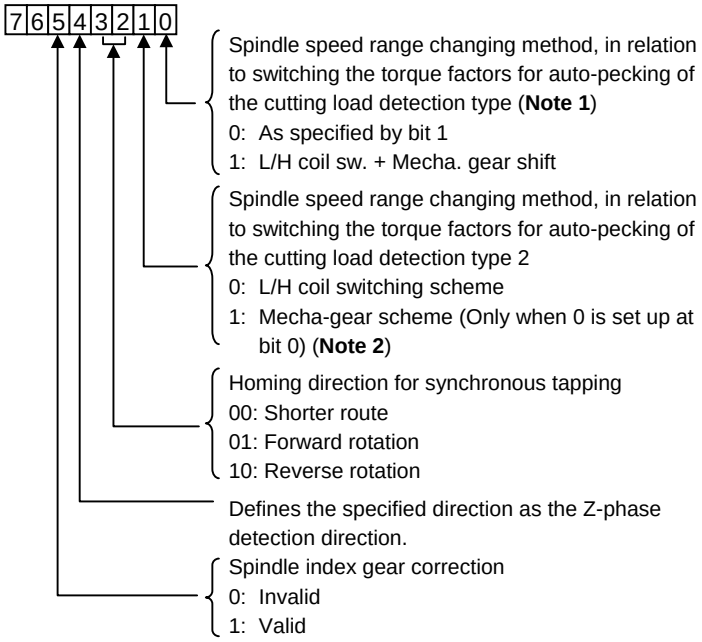
Address	Maximum number of speed ranges			
	1	2	3	4
SA17	○	L	L	L
SA18	Invalid	H	M	ML
SA19	Invalid	Invalid	H	MH
SA20	Invalid	Invalid	Invalid	H

Address	Maximum number of speed range			
	1	2	3	4
SA25	○	L	L	L
SA26	Invalid	H	M	ML
SA27	Invalid	Invalid	H	MH
SA28	Invalid	Invalid	Invalid	H

Classification	MACHINE	Display title	SPINDLE

Address	Meaning		Description
SA42	Minimum spindle speed		Specify the minimum spindle speed.
	Program type	M, E	
	Conditions	At power on	
	Unit	1 min^{-1} (rpm)	
	Setting range	0 to 32767	
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	

Classification	MACHINE	Display title	SPINDLE

Address	Meaning	Description
SA45	Spindle speed range changing method, in relation to switching the torque factors for auto-pecking of the cutting load detection type	

Program type	M, E
Conditions	After stop of movement
Unit	Bit
Setting range	Binary, eight digits

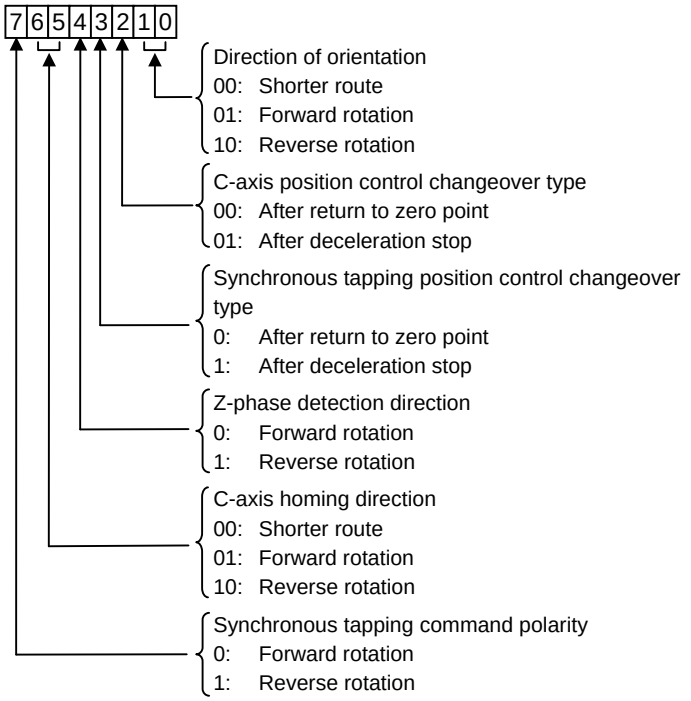
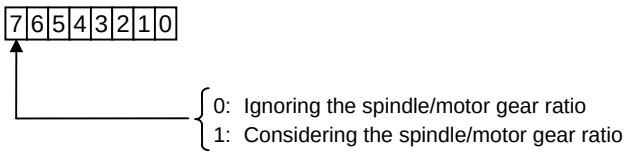
Notes:

1. For the setting of **SA45** bit 0 = 1, set the rated torque, viscous & coulomic friction coefficients of the spindle motor in the relevant parameters as tabulated below:

Speed range No.	Coil	Rated torque	Visc. fric. coef.	Coul. fric. coef.
1	L	**SA53**	**SA57**	**SA61**
1	H	**SA54**	**SA57**	**SA61**
2	L	**SA53**	**SA58**	**SA62**
2	H	**SA54**	**SA58**	**SA62**

2. Bit 1 is valid only when bit 0 = 0.

Classification	MACHINE	Display title	SPINDLE

Address	Meaning		Description
SA46	—		
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	Binary, eight digits	
SA47	—		
	Program type	M, E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	Binary, eight digits	
SA48	Encoder signal input destination		Specify the encoder signal input destination. 0: Via the HDLC-connected axis (Spindle AMP feedback data) 1: Direct connection to encoder 1 (ENC1) 2: Direct connection to encoder 2 (ENC2)
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 2	

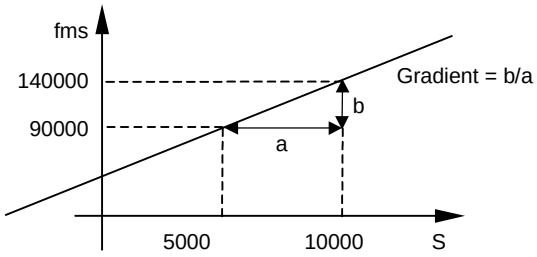
Classification	MACHINE	Display title	SPINDLE

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	1 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	

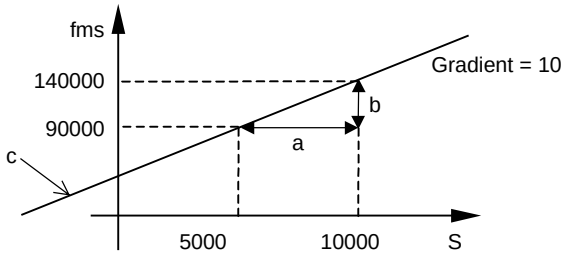
Classification	MACHINE	Display title	SPINDLE

Address	Meaning		Description
SA53	Spindle 1/4h (1/2h) rated torque for auto-pecking of the cutting load detection type — L coils		Set the 1/4h (1/2h) rated torque for the L coils of the spindle motor. **Notes:** **1.** 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. **2.** 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	
SA54	Spindle 1/4h (1/2h) rated torque for auto-pecking of the cutting load detection type — H coils		Set the 1/4h (1/2h) rated torque for the H coils of the spindle motor. **Notes:** **1.** 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. **2.** If bit 1 in **SA45** is set to 1, set the 1/4h rated torque applied when the spindle motor gear position is 2.
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 N·m	
	Setting range	0 to 65535	
SA55	Spindle 1/4h (1/2h) rated torque for auto-pecking of the cutting load detection type — Spindle gear position 3		Set the 1/4h rated torque applied when the spindle motor gear position is 3. **Notes:** **1.** 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. **2.** 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	
SA56	Spindle 1/4h (1/2h) rated torque for auto-pecking of the cutting load detection type — Spindle gear position 4		Set the 1/4h rated torque applied when the spindle motor gear position is 4. **Notes:** **1.** 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. **2.** 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	

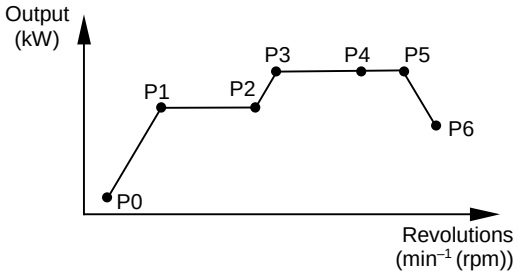
Classification	MACHINE	Display title	SPINDLE

Address	Meaning		Description
SA57	Spindle viscous friction coefficient “cms” for auto-pecking of the cutting load detection type — Range 1		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:  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.)
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	±99999999	
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	
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

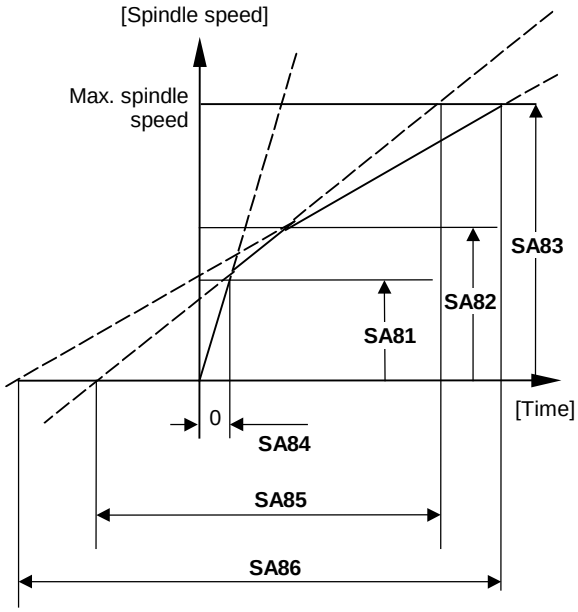
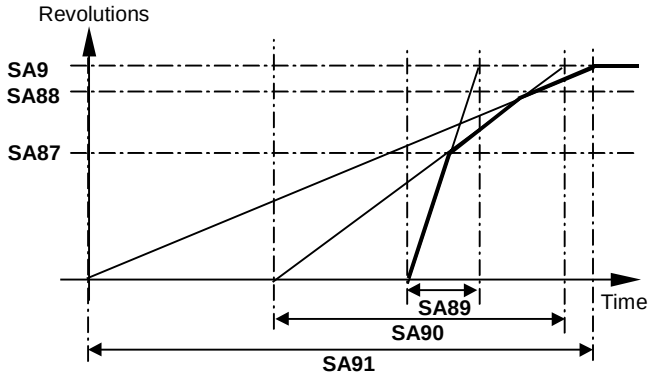
Address	Meaning		Description
SA61	Spindle coulombic friction coefficient “fms” for auto-pecking of the cutting load detection type — Range 1		Set the value where the width of the flat section in the current feedback data matches estimated data. **Example:** If “fms” is 90000 for an “S” value of 5000 and 140000 for S10000, set “c” in **L105**.  Calculate “c” form the linear equation “y = (b/a) x + c”. Since “c” = 90000 – (10 × 5000) = 40000, set “40000” in the above example. **Note:** Set the spindle coulombic friction coefficient “fms” when **SA45** bit 0 = 1.
	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		
	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	

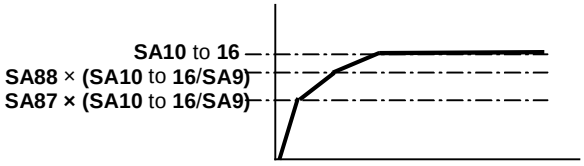
Classification	MACHINE	Display title	SPINDLE

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-msec intervals. If the entered value is “0”, the data actually used will be 4 × 3.5 (msec).
	Program type	M	
	Conditions	Immediate	
	Unit	3.5 msec	
	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	1 min^{-1} (rpm)	
	Setting range	0 to 65535	
SA67 to **SA73**	Revolutions in the following spindle output diagrams: - MACHINING NAVIGATION-RESULT - MACHINING NAVIGATION-PPEDICTION - Monitoring Functions		
	Program type	M, E	
	Conditions	Immediate	
	Unit	1 min^{-1} (rpm)	
	Setting range	0 to 99999999	
SA74 to **SA80**	Output in the following spindle output diagrams: - MACHINING NAVIGATION-RESULT - MACHINING NAVIGATION-PPEDICTION - Monitoring Functions		
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.01 kW	
	Setting range	0 to 99999999	

Point	Cross axis: revolutions (Unit: min^{-1} (rpm))	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**

Classification	MACHINE	Display title	SPINDLE

<table>
<tr><th>Address</th><th colspan="2">Meaning</th><th>Description</th></tr>
<tr><td rowspan="5">SA81
to
SA83</td><td colspan="2">Spindle limit speed selection for spindle position control time constants
SA81: Limit speed 1
SA82: Limit speed 2
SA83: Limit speed 3</td><td rowspan="9">Set the spindle limit speed at which the time constant for the spindle position control is to be changed
 
</td></tr>
<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>1 min^{-1} (rpm)</td></tr>
<tr><td>Setting range</td><td>0 to 99999</td></tr>
<tr><td rowspan="5">SA84
to
SA86</td><td colspan="2">Spindle position control time constants
SA84: Time constant 1
SA85: Time constant 2
SA86: Time constant 3</td></tr>
<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>msec</td></tr>
<tr><td>Setting range</td><td>0 to 99999</td></tr>
<tr><td rowspan="5">SA87
SA88</td><td colspan="2">Spindle speed operating time constant changeover revolutions
SA87: Revolutions 1
SA88: Revolutions 2</td><td rowspan="9"> 

Note:
For gears 2 to 8, SA87 or SA88 is internally converted to calculate the changeover revolutions.
 
</td></tr>
<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>1 min^{-1} (rpm)</td></tr>
<tr><td>Setting range</td><td>0 to 99999</td></tr>
<tr><td rowspan="5">SA89
to
SA91</td><td colspan="2">Spindle speed operating time constant
SA89: Time constant 1
SA90: Time constant 2
SA91: Time constant 3</td></tr>
<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>msec</td></tr>
<tr><td>Setting range</td><td>0 to 9999</td></tr>
</table>

Classification	MACHINE	Display title	SPINDLE

Address	Meaning		Description
SA92	Z-phase detection speed		Set the maximum revolutions for conducting Z-phase detection.
	Program type	M, E	
	Conditions	Immediate	
	Unit	1 min^{-1} (rpm)	
	Setting range	0 to 99999	
SA93	Amount of synchronous tapping zero point shifting		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**).
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	deg	
	Setting range	0 to 359	
SA94	Homing speed for synchronous tapping		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.
	Program type	M, E	
	Conditions	Immediate	
	Unit	1 min^{-1} (rpm)	
	Setting range	0 to 99999	
SA95	Maximum revolutions in manual operation mode		Set the manual operation mode maximum revolutions.
	Program type	M, E	
	Conditions	Immediate	
	Unit	1 min^{-1} (rpm)	
	Setting range	0 to 99999	

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 deg	
	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 = **SA33** × (100 – **SA97**)/100
	Program type	M, E	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 100	
SA98	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
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	Immediate	
	Unit	msec	
	Setting range	0 to 30000	

Classification	MACHINE	Display title	SPINDLE

Address	Meaning		Description
SA100 to **SA113**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
SA114	Spindle speed supervisory mode 2 for safety supervision		Set the spindle speed to be monitored in safety supervisory mode 2. 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.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	1 min^{-1} (rpm)	
	Setting range	0 to 10000	
SA115	Spindle speed supervisory mode 3 for safety supervision		Set the spindle speed to be monitored in safety supervisory mode 3. 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. Assign a value greater than the speed value to be monitored in mode 2 (i.e., specified in **SA114**).
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	1 min^{-1} (rpm)	
	Setting range	0 to 10000	
SA116	Spindle safety clamping mode 2 for safety supervision		Set the spindle safety clamping speed to be monitored in safety supervisory mode 2. In safety supervisory mode 2 and while a safety clamping request is in effect, the spindle is decelerated to the set speed. Assign a value smaller than the speed value to be monitored in mode 2 (i.e., specified in **SA114**).
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	1 min^{-1} (rpm)	
	Setting range	0 to 10000	

Classification	MACHINE	Display title	SPINDLE

Address	Meaning		Description
SA117	Spindle safety clamping mode 3 for safety supervision		Set the spindle safety clamping speed to be monitored in safety supervisory mode 3. In safety supervisory mode 3 and while a safety clamping request is in effect, the spindle is decelerated to the set speed. Assign a value smaller than the speed value to be monitored in mode 3 (i.e., specified in **SA115**). Assign a value greater than the speed value to be monitored in mode 2 (i.e., specified in **SA116**).
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	1 min^{-1} (rpm)	
	Setting range	0 to 10000	
SA118	Selecting the spindle door of the spindle whose speed is to be monitored		Select the door group to which the spindle belongs in safety supervisory mode. If SP129: SFNC9/bit F is OFF, speed monitoring will not be conducted, irrespective of the setting of this parameter. [Examples of setting] 0000: The spindle does not belong to any door. 0001: The spindle belongs to the door-1 group. 0002: The spindle belongs to the door-2 group. 0003: The spindle belongs to the door-1/2 group.
	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		For safety speed clamping in safety supervisory mode, when a claming 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. 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).
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	%	
	Setting range	0 to 100	
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	

Classification	MACHINE	Display title	SPINDLE

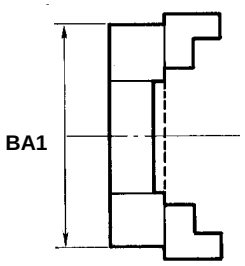
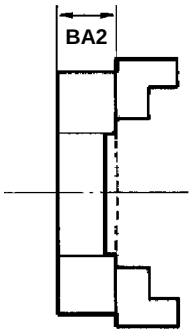
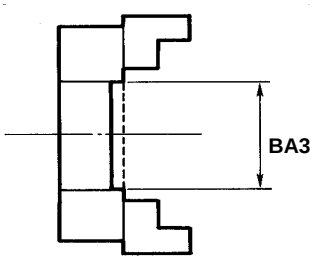
Address	Meaning		Description
SA121 to **SA128**	Amount of branching point correction for spindle index gear tooth **SA121**: Branching point [1] **SA122**: Branching point [2] **SA123**: Branching point [3] **SA124**: Branching point [4] **SA125**: Branching point [5] **SA126**: Branching point [6] **SA127**: 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 deg	
	Setting range	–99999999 to 99999999	
SA129 to **SA137**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
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	
SA140	Turret indexing gear ratio		Set the gear ratio for turret indexing.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	—	
	Setting range	—	

Classification	MACHINE	Display title	SPINDLE

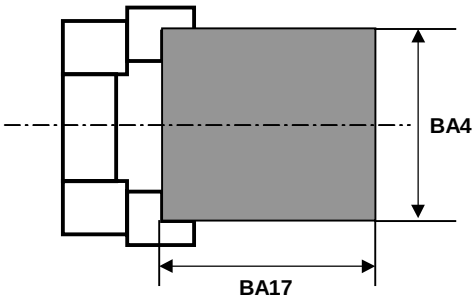
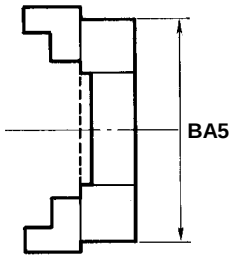
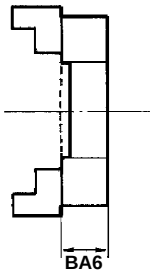
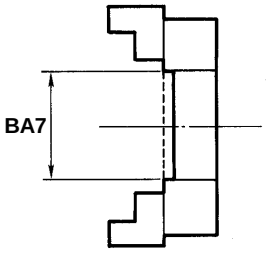
Address	Meaning		Description
SA141 to **SA143**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
SA144	—		7 6 5 4 3 2 1 0 Spindle gear changeover valid Turret indexing valid.
	Program type	M, E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	Binary, eight digits	

2-3-16 Machine parameter BARRIER (BA)

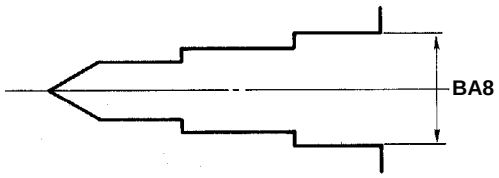
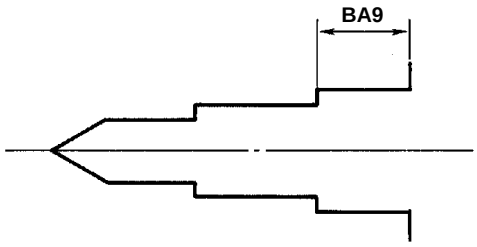
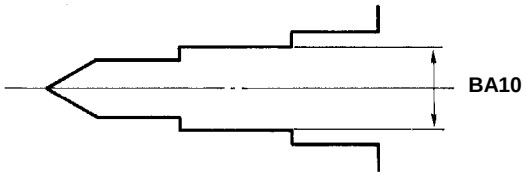
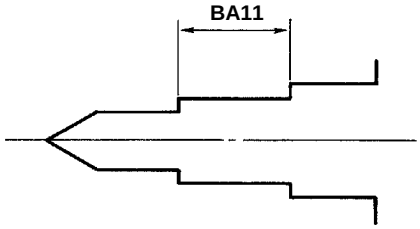
Classification	MACHINE	Display title	BARRIER

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 inch	
	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 inch	
	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 inch	
	Setting range	0 to 99999999	

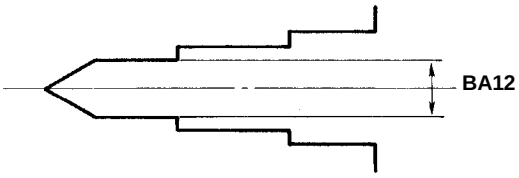
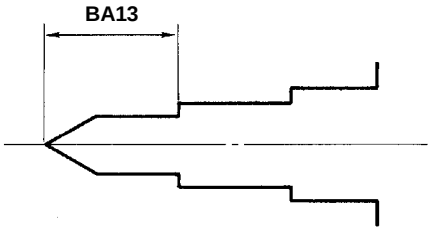
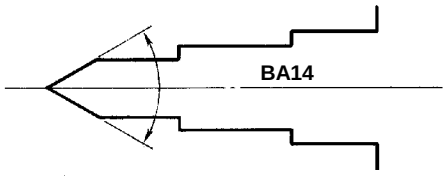
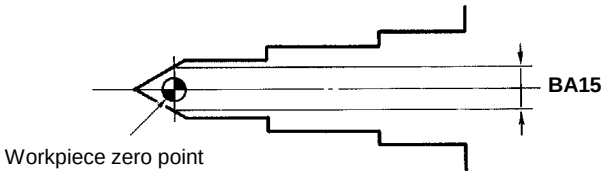
Classification	MACHINE	Display title	BARRIER

Address	Meaning		Description
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 inch	
	Setting range	0 to 99999999	
BA5	Chuck outside diameter (for chuck barrier) — No. 2 turning spindle		Setting of chuck outside diameter of the No. 2 turning spindle  NM211-00312 **Note:** For single turning-spindle specifications, this parameter is invalid.
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 99999999	
BA6	Chuck width (for chuck barrier) — No. 2 turning spindle		Setting of chuck width of the No. 2 turning spindle  NM211-00313 **Note:** For single turning-spindle specifications, this parameter is invalid.
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 99999999	
BA7	Chuck inside diameter (for chuck barrier) — No. 2 turning spindle		Setting of chuck inside diameter of the No. 2 turning spindle  NM211-00314 **Note:** For single turning-spindle specifications, this parameter is invalid.
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 99999999	

Classification	MACHINE	Display title	BARRIER

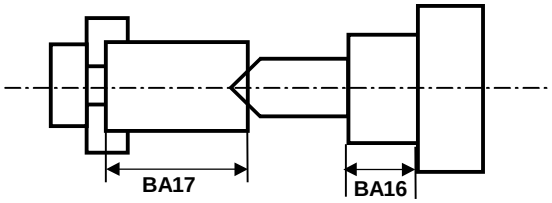
Address	Meaning		Description
BA8	Tail body outside diameter (for tail barrier)		Setting of tail body outside diameter  NM211-00315
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 99999999	
BA9	Tail body length (for tail barrier)		Setting of tail body length  NM211-00316
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 99999999	
BA10	Tail spindle outside diameter (for tail barrier)		Setting of tail spindle outside diameter  NM211-00317
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 99999999	
BA11	Length with tail spindle at back end (for tail barrier)		Setting of length with tail spindle at back end  NM211-00318
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 99999999	

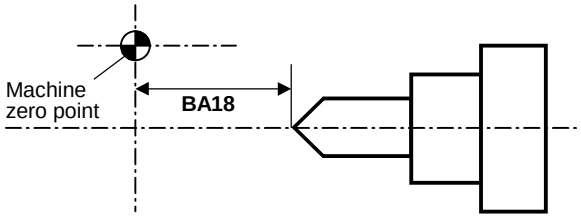
Classification	MACHINE	Display title	BARRIER

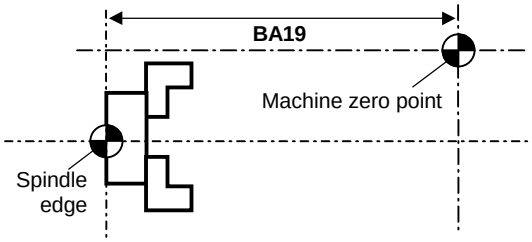
Address	Meaning		Description
BA12	Tail head outside diameter (for tail barrier)		Setting of tail head outside diameter  NM211-00319
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 99999999	
BA13	Tail head length (for tail barrier)		Setting of tail head length  NM211-00320
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 99999999	
BA14	Tail head taper angle (for tail barrier)		Setting of tail head taper angle  NM211-00321
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 deg	
	Setting range	0 to 180000	
BA15	Tail head biting diameter (for tail barrier)		Setting of biting diameter when tail head is used  NM211-00322
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 99999999	

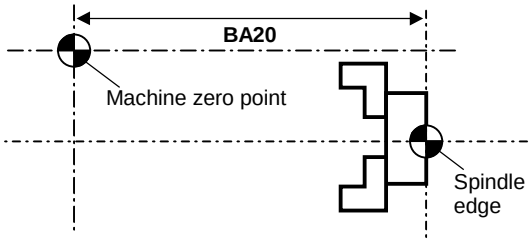
Classification	MACHINE

Display title	BARRIER

<table>
<tr><th>Address</th><th colspan="2">Meaning</th><th>Description</th></tr>
<tr><td rowspan="5">BA16
to
BA18</td><td colspan="2" rowspan="2">EIA tail barrier
Tail dimensions</td><td rowspan="5">Set the dimensional data for forming tail barriers using an EIA/ISO program.
<During tail operation>
 

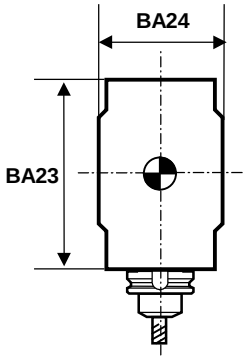
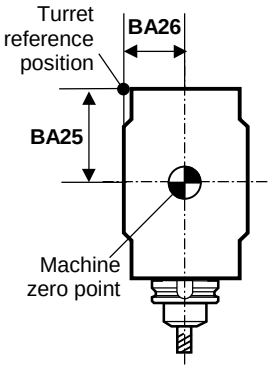
<During tail reversing>
 

BA16: Tail extruding length
BA17: Workpiece length
BA18: Distance from the machine zero point to the leading edge position during tail reversing
Notes:
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.</td></tr>
<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 inch</td></tr>
<tr><td>Setting range</td><td>0 to 99999999 (Note 1)</td></tr>
<tr><td rowspan="5">BA19</td><td colspan="2" rowspan="2">Distance from the Z-axis machine zero point to the spindle edge
— No. 1 turning spindle</td><td rowspan="5">Specify the distance from the machine zero point of the Z-axis to the edge of the No. 1 turning spindle.
 

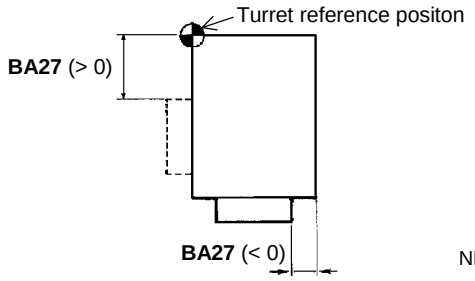
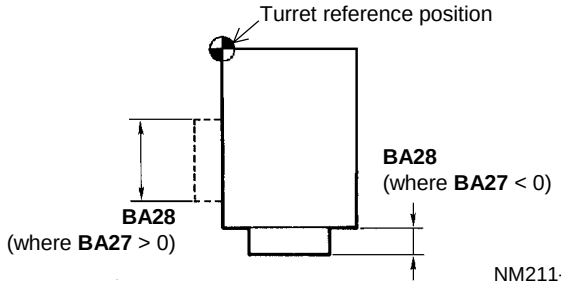
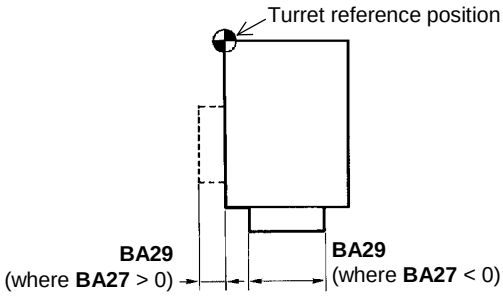
Note:
Enter the distance with minus sign for the machine zero point in the minus direction with respect to the spindle edge.</td></tr>
<tr><td>Program type</td><td>M, E</td></tr>
<tr><td>Conditions</td><td>Immediate</td></tr>
<tr><td>Unit</td><td>0.001 mm/0.0001 inch</td></tr>
<tr><td>Setting range</td><td>±99999999</td></tr>
<tr><td rowspan="5">BA20</td><td colspan="2" rowspan="2">Distance from the Z-axis machine zero point to the spindle edge
— No. 2 turning spindle</td><td rowspan="5">Specify the distance from the machine zero point of the Z-axis to the edge of the No. 2 turning spindle.
 

Note:
Enter the distance with minus sign for the machine zero point in the minus direction with respect to the spindle edge.</td></tr>
<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 inch</td></tr>
<tr><td>Setting range</td><td>±99999999</td></tr>
</table>

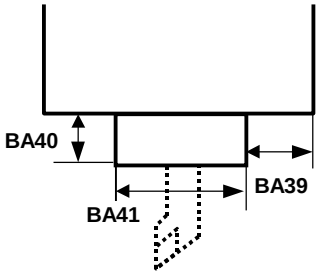
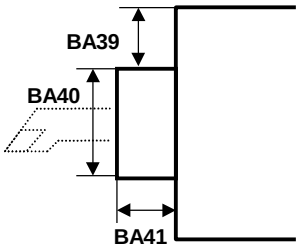
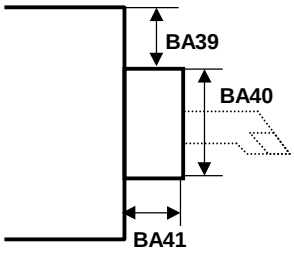
Classification	MACHINE	Display title	BARRIER

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.  **BA23:** Outline of the turret **BA24:** Width of the turret
	Program type	E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 99999999	
BA25 **BA26**	Turret reference position		Specify the turret reference position from the machine zero point.  **BA23:** Turret reference position X **BA24:** Turret reference position Z
	Program type	E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	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		Setting of tool holder mounting position. When plus data is used, the tool holder is mounted horizontally, and minus data downward. **Example:** Type 1  NM211-00327' Same for types 2, 3, 4
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	±99999999	
BA28 **BA31** **BA34** **BA37**	Tool holder width in X-axis direction BA28 - Type 1 BA31 - Type 2 BA34 - Type 3 BA37 - Type 4		Setting tool holder width in X-axis direction **Example:** Type 1  NM211-00328' Same for types 2, 3, 4
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 99999999	
BA29 **BA32** **BA35** **BA38**	Tool holder width in Z-axis direction BA29 - Type 1 BA32 - Type 2 BA35 - Type 3 BA38 - Type 4		Setting of tool holder width in Z-axis direction **Example:** Type 1  NM211-00329' Same for types 2, 3, 4
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 99999999	

Classification	MACHINE	Display title	BARRIER

Address	Meaning		Description
BA39 to **BA41**	EIA tool barrier Holder dimensions		Set the holder shape data for forming tool barriers using an EIA/ISO program. <Holder-under type> Input **BA39** with a minus sign.  <Holder-side type (0° type)> Input **BA39** with a plus sign. Input **BA41** with a plus sign.  <Holder-side type (180° type)> Input **BA39** with a plus sign. Input **BA41** with a minus sign.  **BA39:** EIA tool barrier, Holder mounting position **BA40:** EIA tool barrier, X-axial width of the holder **BA41:** EIA tool barrier, Z-axial width of the holder **Note:** 0 to 99999999 for **BA40**.
	Program type	E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	±99999999 (**Note**)	
BA42	Barrier type		Select the type of barrier to be made valid. 0: Type with head B-axis (tool rotational B-axis) [INTEGREX type] 1: Type without head B-axis (tool rotational B-axis)
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 99999999	

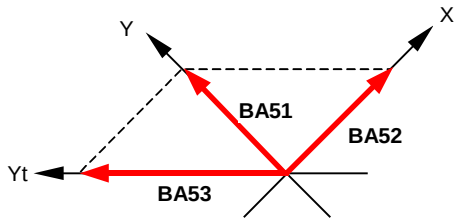
Classification	MACHINE	Display title	BARRIER

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		Set the type of turret: 0: ATC 1: Turret 2: Lower turret 3: Opposed turret 4: Work rest
	Program type	M, E	
	Conditions	At power on	
	Unit	—	
	Setting range	0 to 4	
BA48	Axis name of the head to be rotated		Set the axis name of the head axis to be rotated. Set “&42” if the head axis to be rotated is the B-axis.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	—	
	Setting range	&41 to &5A	
BA49	Axis number of the inclined axis (Inclined-axis control)		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.
	Program type	M, E	
	Conditions	At power on	
	Unit	—	
	Setting range	0 to 16	
BA50	Fundamental axis number (Inclined-axis control)		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.
	Program type	M, E	
	Conditions	At power on	
	Unit	—	
	Setting range	0 to 16	

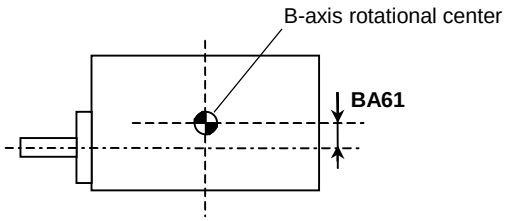
Classification	MACHINE	Display title	BARRIER

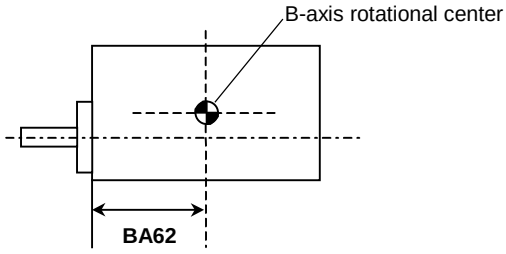
Address	Meaning		Description
BA51	Vector of virtual Y (Inclined-axis control)		Specify one of the vectors created by the triangles formed by inclination angles. **BA51:** Vector of the inclined axis (virtual Y) in the rectangular coordinate system **BA52:** Vector of the fundamental axis (X-axis) corresponding to the inclined axis **BA53:** Vector of the real axis corresponding to the inclined axis  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	
BA52	Vector of real X (Inclined-axis control)		
	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		Set the axis number of the work spindle to be used for hobbing. The hobbing is invalid when this parameter is set to 0.
	Program type	E	
	Conditions	After stop of movement	
	Unit	—	
	Setting range	0 to 16	

Classification	MACHINE	Display title	BARRIER

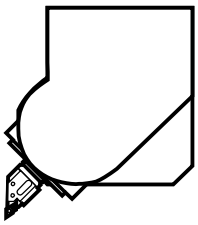
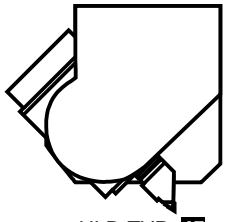
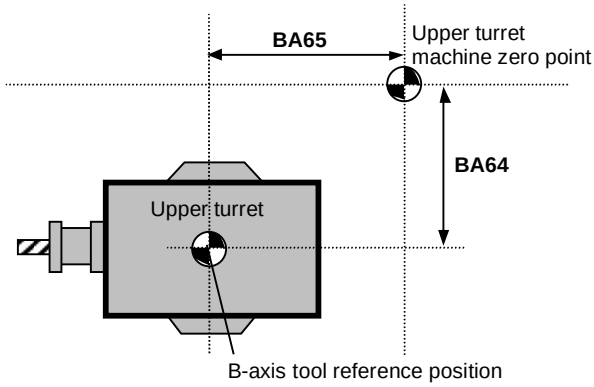
Address	Meaning		Description
BA55	Turning spindle number for polygonal machining (D1)		Set the turning spindle to be used in the D1 command mode of polygonal machining. 0: Turning spindle No. 1 1: Turning spindle No. 2 2: Turning spindle No. 3 3: Turning spindle No. 4 –1: Invalid
	Program type	E	
	Conditions	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

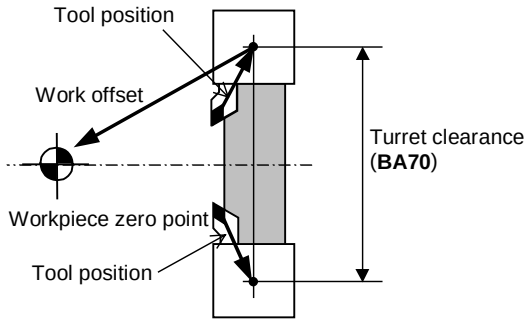
<table>
<tr><th>Address</th><th colspan="2">Meaning</th><th>Description</th></tr>
<tr><td rowspan="5">BA59</td><td colspan="2">Spindle forward rotation M-code for tapping cycle</td><td rowspan="10">- If the command block of G284/G288 (machining G-codes, F30 = 0) or of G84/G88 (turning G-codes, F30 = 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>
<tr><td colspan="2"></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>
- 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.</td></tr>
<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 255</td></tr>
<tr><td rowspan="5">BA60</td><td colspan="2">Spindle reverse rotation M-code for tapping cycle</td></tr>
<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>
<tr><td rowspan="5">BA61</td><td colspan="2">Amount of runout of the B-axis center</td><td rowspan="5">Enter the distance from the rotational center of the B-axis to the tool center.
 

Note:
Enter the distance with a minus sign for B-axis rotational center above the tool center position.</td></tr>
<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</td></tr>
<tr><td>Setting range</td><td>–1000 to 1000</td></tr>
<tr><td rowspan="5">BA62</td><td colspan="2">Amount of offset for the B-axis — spindle distance</td><td rowspan="5">Enter the distance from the rotational center of the B-axis to the spindle edge.
 
</td></tr>
<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 inch</td></tr>
<tr><td>Setting range</td><td>±99999999</td></tr>
</table>

Classification	MACHINE	Display title	BARRIER

Address	Meaning		Description
BA63	Holder angle of angle tool holder		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. **BA63** = 45  HLD.TYP: 45 (**BA63** = 45)  HLD.TYP: 45 (180 – **BA63** = 135)
	Program type	M	
	Conditions		
	Unit	1°	
	Setting range	0 to 90	
BA64	B-axis tool reference position X		
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	±99999999	
BA65	B-axis tool reference position Z		
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	±99999999	
BA66 **BA67**	G37 deceleration area G37 measuring area		Set the deceleration area and measuring area in the G37 command. **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. **BA67**: Set the moving distance in measuring feed mode. This value will be used when argument D is omitted from the G37 command.
	Program type	E	
	Conditions	After stop of movement	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 99999999	

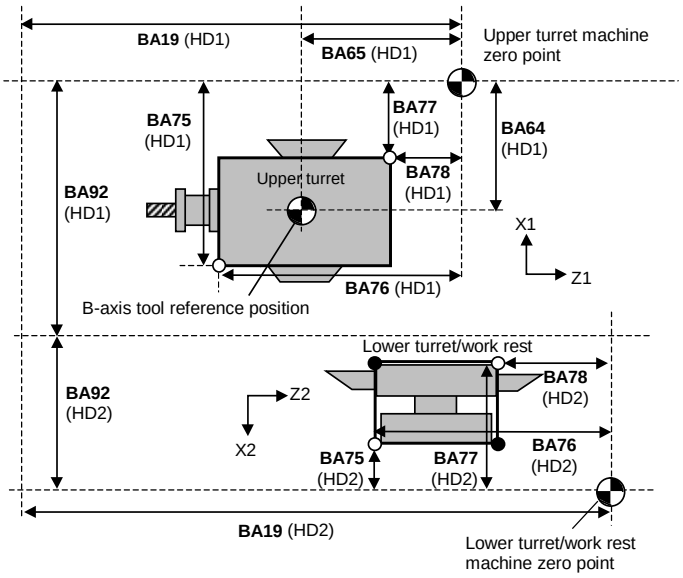
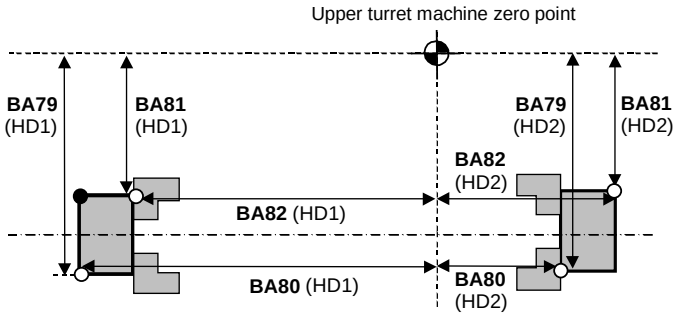
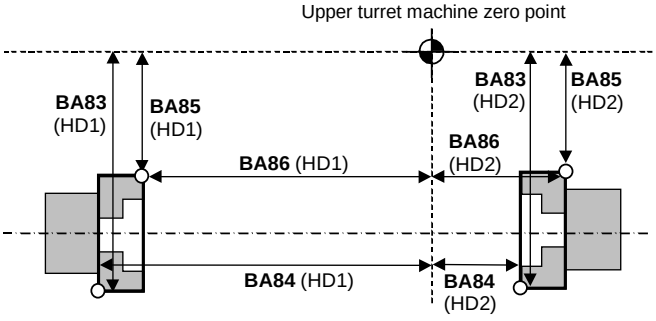
Classification	MACHINE	Display title	BARRIER

Address	Meaning		Description
BA68 **BA69**	G36 deceleration area G36 measuring area		Set the deceleration area and measuring area in the G36 command. **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. **BA69**: Set the moving distance in measuring feed mode. This value will be used when argument D is omitted from the G36 command.
	Program type	E	
	Conditions	After stop of movement	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 99999999	
BA70	Distance between the reference points on both turrets (radius value)		- 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 <Schematic diagram> 
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	0 to 99999999	
BA71	System number to be used when argument L is omitted from G112		0: System 1 1: System 2 2: System 3 3: System 4
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 3	

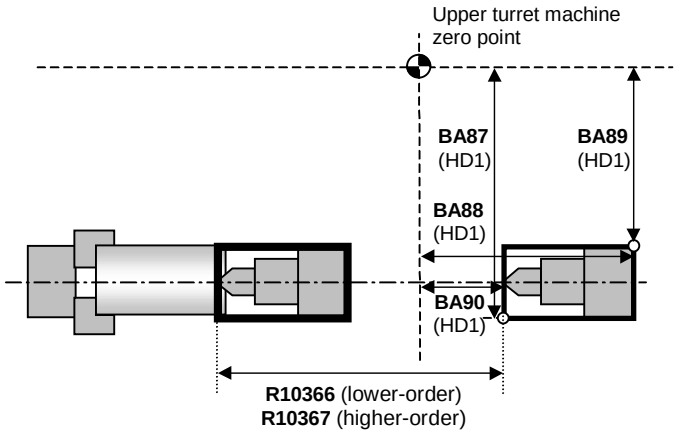
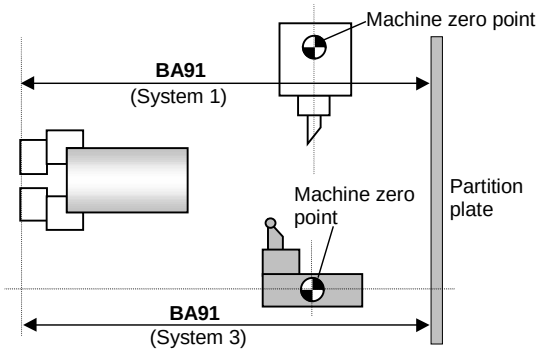
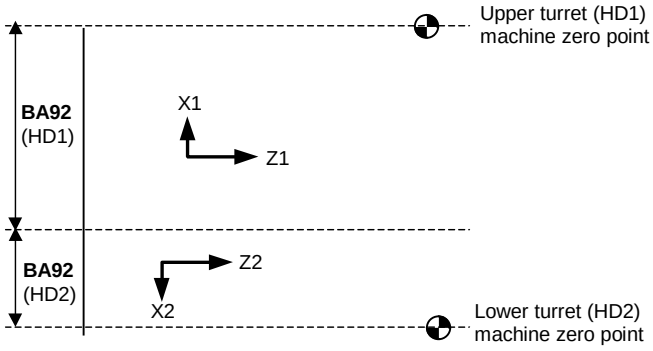
Classification	MACHINE	Display title	BARRIER

Address	Meaning		Description
BA72	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
BA73	Barrier valid/invalid 1 (chuck, sub-chuck, tailstock)		0: Chuck, sub-chuck and tailstock invalid 1: Chuck valid 2: Sub-chuck valid 3: Tailstock valid
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 3	
BA74	Barrier valid/invalid 2 (Lower turret, work rest)		0: Lower turret and work rest invalid 1: Lower turret valid 2: Work rest valid
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 2	

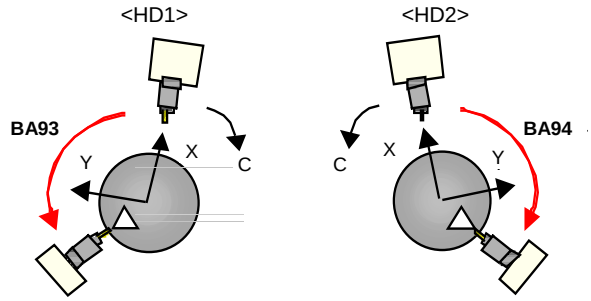
Classification	MACHINE	Display title	BARRIER

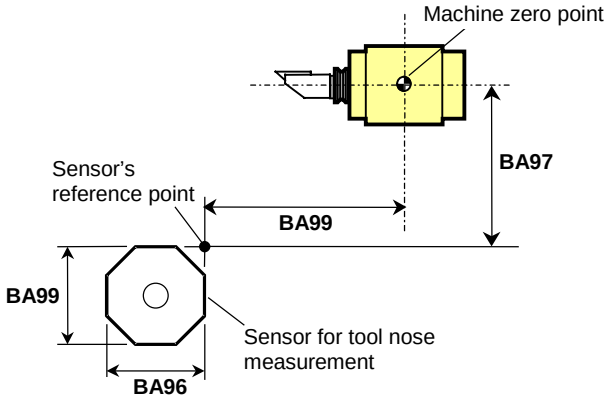
Address	Meaning		Description
BA75 to **BA78**	Barrier setup turret reference position		Set the reference position when it is viewed from the machine zero point. 
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	±99999999	
BA79 to **BA82**	Barrier setup chuck reference position 1		Set the reference position when it is viewed from the machine zero point. 
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	±99999999	
BA83 to **BA86**	Barrier setup chuck reference position 2		Set the reference position within a jaw barrier area not having a specified jaw number. 
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	±99999999	

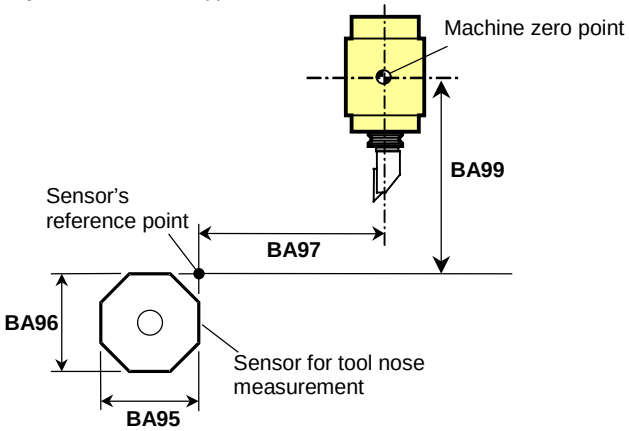
Classification	MACHINE	Display title	BARRIER

Address	Meaning		Description
BA87 to **BA90**	Tail barrier reference position		Set the reference position within a tail barrier area. The reference position here refers to the position when viewed from the machine zero point of the upper turret of the tailstock being not used for machining. 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. Only parameters for HD1 are used. 
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	±99999999	
BA91	Distance from spindle edge to partition plate		Set the reference position of the partition plate. 
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range	±99999999	
BA92	Central position X when viewed from machine zero point (Barrier function)		Set the central position X when viewed from the machine zero point. 
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 inch	
	Setting range		

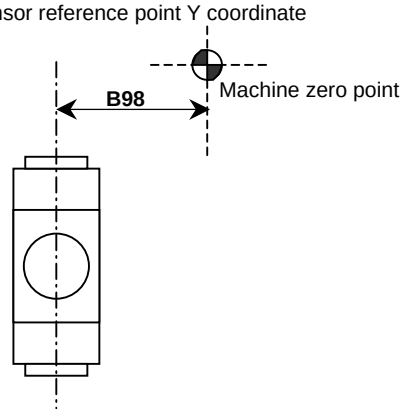
Classification	MACHINE	Display title	BARRIER

<table>
<tr><th>Address</th><th colspan="2">Meaning</th><th>Description</th></tr>
<tr><td>BA93
BA94</td><td colspan="2">BA93: Upper/lower turret tool angle difference (HD1)
BA94: Upper/lower turret tool angle difference (HD2)</td><td rowspan="5">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.
 
</td></tr>
<tr><td rowspan="4"></td><td>Program type</td><td>M, E</td></tr>
<tr><td>Conditions</td><td>Immediate</td></tr>
<tr><td>Unit</td><td>0.0001 deg</td></tr>
<tr><td>Setting range</td><td>0 to ±99999999</td></tr>
<tr><td>BA95
BA96
BA97
BA99
BA100
BA102
BA103
BA104</td><td colspan="2">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

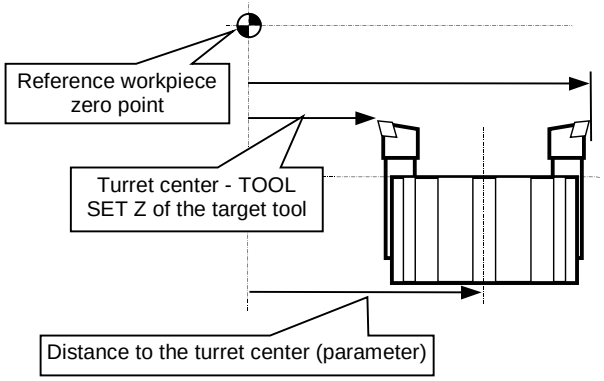
BA103: Sensor width along the X-axis (for lower turret/HD2)
BA104: Sensor width along the Z-axis (for lower turret/HD2)
BA100: X-coordinate of the sensor’s reference point (for lower turret/HD2)
BA102: Z-coordinate of the sensor’s reference point (for lower turret/HD2)</td><td rowspan="5">Use BA95 and 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
 

Example 2: For vertical type
 
</td></tr>
<tr><td rowspan="4"></td><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 inch</td></tr>
<tr><td>Setting range</td><td>±99999999</td></tr>
</table>

Classification	MACHINE	Display title	BARRIER

Address	Meaning		Description
BA98 **BA101**	**BA98**: Y-coordinate of the sensor's reference point **BA101**: Y-coordinate of the sensor's reference point (for lower turret/HD2)		Setting of sensor reference point Y coordinate 
	Program type	M	
	Conditions	Immediate	
	Unit	0.0001 mm/0.00001 inch	
	Setting range	±99999999	
BA105	Adjustment "FdT" for W-axis thrust hold		Set adjustment "FdT" for holding the thrust of the W-axis. Set a value less than **BA108** × **BA106**/100. If any other value is set, the thrust may be unstable.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	0.0005 mm/0.000005 inch	
	Setting range	0 to 99999999	
BA106	Dead zone		Set the dead zone for the drooping level of the W-axis thrust. A drooping level change less than **BA108** × **BA106**/100 is ignored.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	%	
	Setting range	0 to 99999999	
BA107	Filter		Set the monitoring time for changes in the drooping level of the W-axis thrust.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	1/3.5 msec	
	Setting range	0 to 99999999	

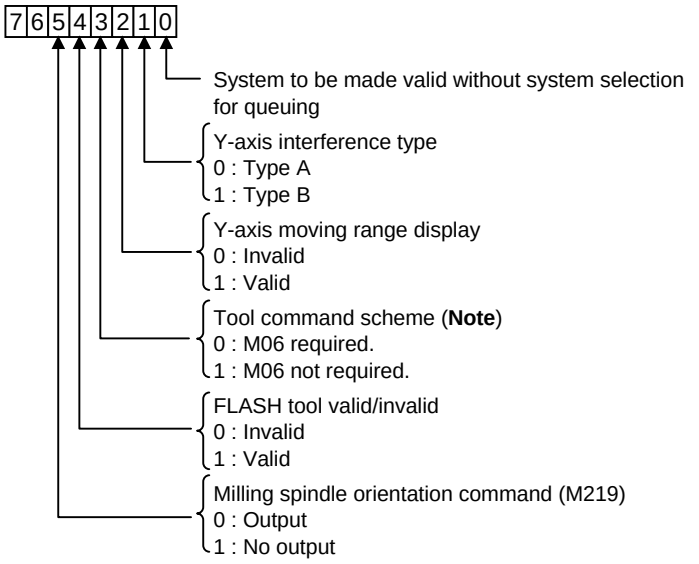
Classification	MACHINE	Display title	BARRIER

Address	Meaning		Description
BA108	W-axis thrust hold droop		Set the drooping level for holding the W-axis thrust. This drooping level is held during the application of the W-axis thrust.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	0.0005 mm/0.000005 inch	
	Setting range	0 to 99999999	
BA109	Offset amount (X) during automatic tool setting value calculation		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 oppsite position. 
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	±99999999	
BA110	Offset amount (Z) during automatic tool setting value calculation		TOOL SET X = Tool setting value X for target tool + Parameter for offset amount X TOOL SET Z = (Parameter for distance to center of turret × (–1) × 2) – Tool setting value Z for target tool + Parameter for offset amount Z
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	±99999999	
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		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.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	mm/min	
	Setting range	0 to 999999	

Classification	MACHINE	Display title	BARRIER

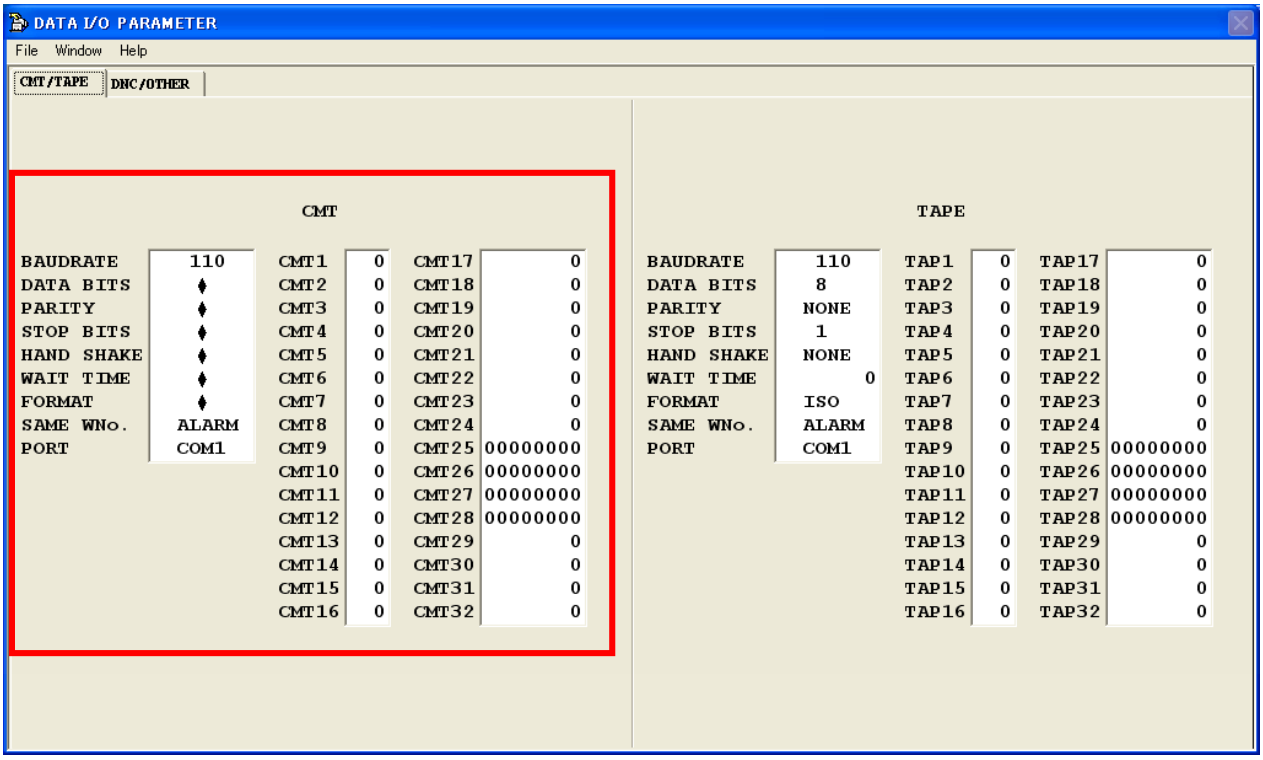
<table>
<tr><th>Address</th><th colspan="2">Meaning</th><th>Description</th></tr>
<tr><td rowspan="5">BA117
to
BA124</td><td colspan="2">—</td><td>Invalid</td></tr>
<tr><td>Program type</td><td>—</td><td rowspan="4"></td></tr>
<tr><td>Conditions</td><td>—</td></tr>
<tr><td>Unit</td><td>—</td></tr>
<tr><td>Setting range</td><td>—</td></tr>
<tr><td rowspan="5">BA125</td><td colspan="2" rowspan="4">EIA barrier activation</td><td>Set the types of tailstock, tool holder and chuck for which the EIA barriers are to be activated.
7 6 5 4 3 2 1 0
Tailstock usage valid/invalid (Note 1)
0: Invalid
1: Valid
Tailstock present/absent (Note 2)
0: Absent
1: Present
Tool holder present/absent
0: Absent
1: Present
Barrier tool nose position
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
0: Invalid
1: Valid
Notes:
1. “Tailstock usage invalid” does not mean cancelling 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.</td></tr>
<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>Binary, eight digits</td></tr>
</table>

Classification	MACHINE	Display title	BARRIER

Address	Meaning		Description
BA126	—		 **Note:** T-command operation is defined as follows by the setting of bit 3: - If the setting of bit 3 is 0 TO**T**◆M6: Replaces the current tool with O 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 TO□: Replaces the current tool with O and set □ as the tool offset number.
	Program type	E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	Binary, eight digits	
BA127 to **BA132**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

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 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: 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 Set values / Description COM1 / CF22 serial ch1 COM2 / CF22 serial ch2 COM3 / Invalid COM4 / Invalid
Program type	M, E	
Conditions	At I/O startup	
Unit	—	
Setting range	—	

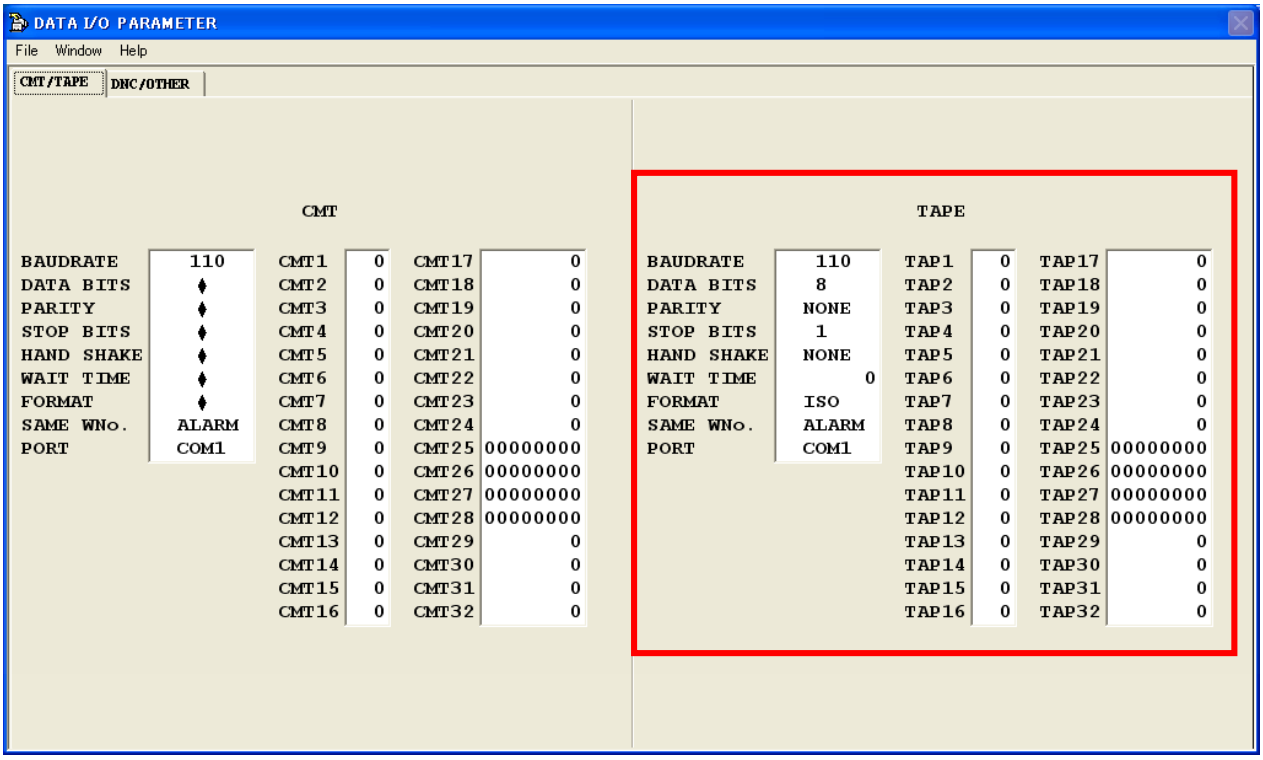
Classification	DATA I/O

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 Set values 110 / 4800 300 / 9600 1200 / 19200 2400
Program type	M, E	
Conditions	At I/O startup	
Unit	—	
Setting range	110 to 19200	
DATA BITS		Number of data bits (parameter for RS-232C interface initialization) Set values 7 8
Program type	M, E	
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 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 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. Set values: NONE — No control Set values: DC CONTROL — Complies with control code DC1 through DC4 Set values: 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 sec.	
Setting range	0 to 65535	

Classification	DATA I/O

Display title	TAPE

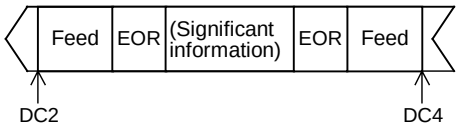
Meaning		Description
FORMAT		Selection of paper tape puncher output code Set values: ISO — Paper tape punching in ISO code Set values: EIA — Paper tape punching in EIA code Set values: ASCII — Paper tape punching in ASCII code
Program type	M, E	
Conditions	At I/O startup	
Unit	—	
Setting range	—	
SAME WNo.		Type of processing to be executed if the machining program of an existing work number is to be loaded Set values: ALARM — Issues an alarm if the received work number already exists. Set values: 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 Set values: COM1 — CF22 serial ch1 Set values: COM2 — CF22 serial ch2 Set values: COM3 — Invalid Set values: COM4 — Invalid
Program type	M, E	
Conditions	At I/O startup	
Unit	—	
Setting range	—	

Classification	DATA I/O

Display title	TAPE

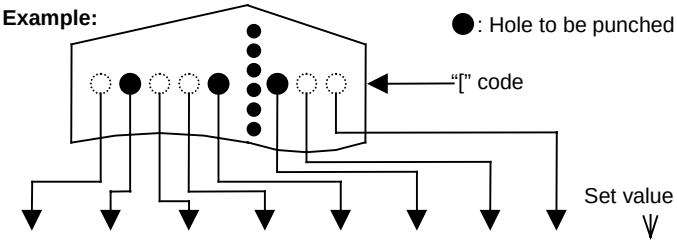
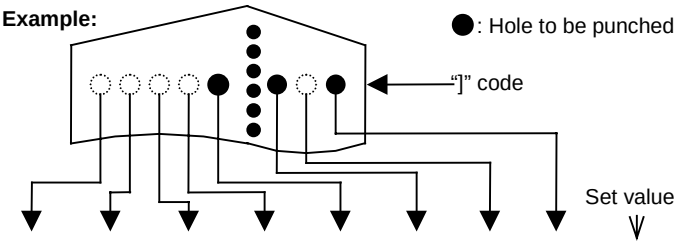
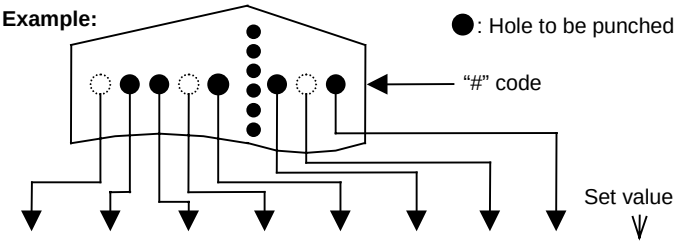
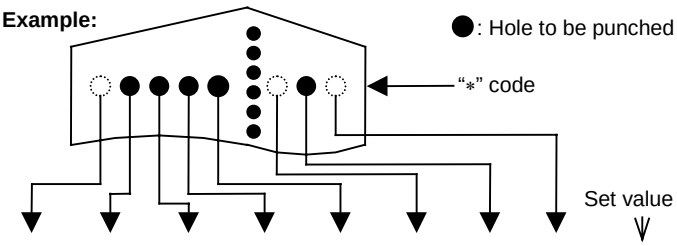
Address	Meaning		Description
TAP1	Type of terminator		Set values / Terminator: 0 Without terminator; 1 EOB or EOR; 2 EOB only; 3 EOR only; 4 One character of your choice; 5 Two characters of your choice
	Program type	M, E	
	Conditions	At I/O startup	
	Unit	—	
	Setting range	0 to 5	
TAP2	Terminator code 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		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. 0: No placement of CR 1: Placement of CR
	Program type	M, E	
	Conditions	At I/O startup	
	Unit	—	
	Setting range	0, 1	

Classification	DATA I/O	Display title	TAPE

<table>
<tr><th>Address</th><th colspan="2">Meaning</th><th>Description</th></tr>
<tr><td rowspan="5">TAP5</td><td colspan="2">DC code parity</td><td rowspan="5">This parameter is used to specify whether or not a parity bit is to be assigned to the DC code to be output.
<table><tr><th>Set values</th><th>Parity</th><th colspan="10">Hole-punching pattern of DC3 code</th></tr><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></table>Note:
This parameter is valid only when HAND SHAKE is set to DC CONTROL.</td></tr>
<tr><td>Program type</td><td>M, E</td></tr>
<tr><td>Conditions</td><td>At I/O startup</td></tr>
<tr><td>Unit</td><td>—</td></tr>
<tr><td>Setting range</td><td>0, 1</td></tr>
<tr><td rowspan="5">TAP6</td><td colspan="2">Feed section DC code output</td><td rowspan="5">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.
Example:
 

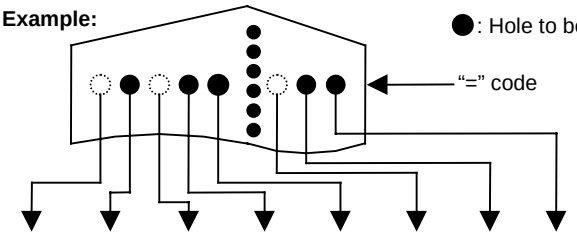
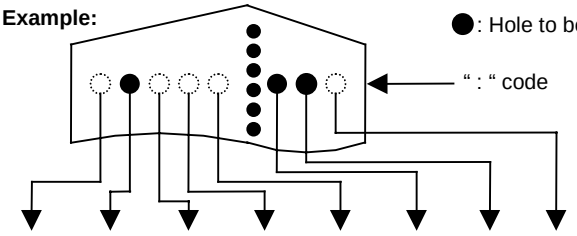
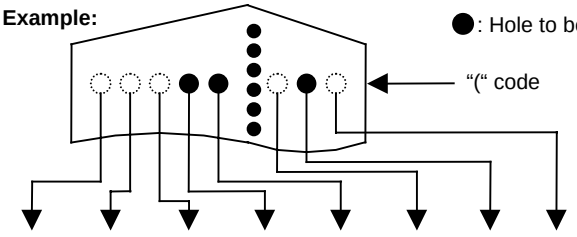
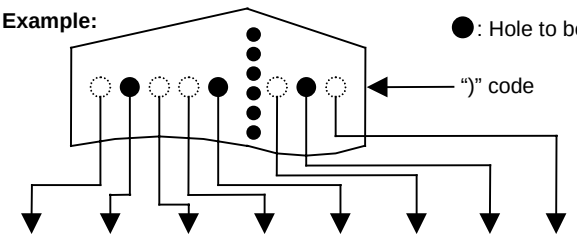
<table><tr><th>Set values</th><th>Description</th></tr><tr><td>0</td><td>Neither DC2 nor DC4 is output.</td></tr><tr><td>1</td><td>Only DC2 is output.</td></tr><tr><td>2</td><td>Only DC4 is output.</td></tr><tr><td>3</td><td>Both DC2 and DC4 are output.</td></tr></table>Note:
This parameter is valid only when HAND SHAKE is set to DC CONTROL.</td></tr>
<tr><td>Program type</td><td>M, E</td></tr>
<tr><td>Conditions</td><td>At I/O startup</td></tr>
<tr><td>Unit</td><td>—</td></tr>
<tr><td>Setting range</td><td>0 to 3</td></tr>
<tr><td rowspan="5">TAP7
TAP8</td><td colspan="2">—</td><td rowspan="5">Invalid</td></tr>
<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>

Classification	DATA I/O	Display title	TAPE

Address	Meaning		Description
TAP9	"[" code for paper tape reader/puncher for EIA		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. **Example:** ●: Hole to be punched  $(0 \times 2^7)+(1 \times 2^6)+(0 \times 2^5)+(0 \times 2^4)+(1 \times 2^3)+(1 \times 2^2)+(0 \times 2^1)+(0 \times 2^0) = \underline{76}$ MPL068
	Program type	M, E	
	Conditions	At I/O startup	
	Unit	—	
	Setting range	0 to 255	
TAP10	"]" code for paper tape reader/puncher for EIA		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. **Example:** ●: Hole to be punched  $(0 \times 2^7)+(0 \times 2^6)+(0 \times 2^5)+(0 \times 2^4)+(1 \times 2^3)+(1 \times 2^2)+(0 \times 2^1)+(1 \times 2^0)=\underline{13}$ MPL069
	Program type	M, E	
	Conditions	At I/O startup	
	Unit	—	
	Setting range	0 to 255	
TAP11	"#" code for paper tape reader/puncher for EIA		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. **Example:** ●: Hole to be punched  $(0 \times 2^7)+(1 \times 2^6)+(1 \times 2^5)+(0 \times 2^4)+(1 \times 2^3)+(1 \times 2^2)+(0 \times 2^1)+(1 \times 2^0)=\underline{109}$ MPL070
	Program type	M, E	
	Conditions	At I/O startup	
	Unit	—	
	Setting range	0 to 255	
TAP12	"∗" code for paper tape reader/puncher for EIA		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. **Example:** ●: Hole to be punched  $(0 \times 2^7)+(1 \times 2^6)+(1 \times 2^5)+(1 \times 2^4)+(1 \times 2^3)+(0 \times 2^2)+(1 \times 2^1)+(0 \times 2^0)=\underline{122}$ MPL071
	Program type	M, E	
	Conditions	At I/O startup	
	Unit	—	
	Setting range	0 to 255	

Classification	DATA I/O

Display title	TAPE

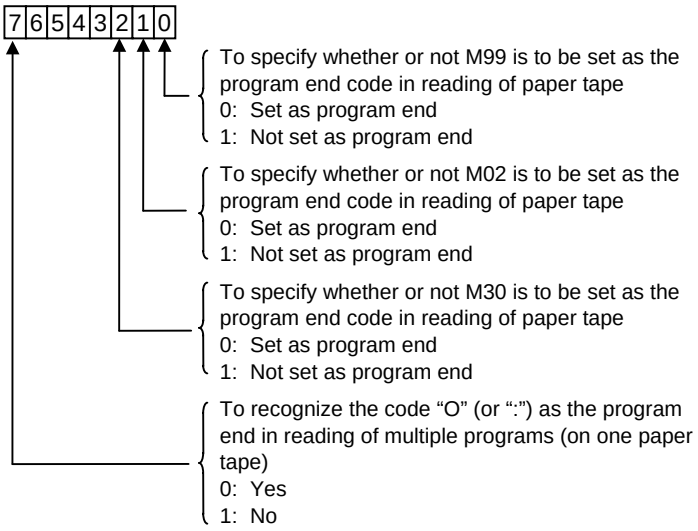
Address	Meaning		Description
TAP13	"=" code for paper tape reader/puncher for EIA		This parameter is used to set a hole-punching pattern for the character code "=" onto a paper tape reader/puncher using EIA. Set an eight-digit binary number in decimal form. **Example:** ●: Hole to be punched  $(0 \times 2^7)+(1 \times 2^6)+(0 \times 2^5)+(1 \times 2^4)+(1 \times 2^3)+(0 \times 2^2)+(1 \times 2^1)+(1 \times 2^0)=\underline{91}$ MPL072
	Program type	M, E	
	Conditions	At I/O startup	
	Unit	—	
	Setting range	0 to 255	
TAP14	":" code for paper tape reader/puncher for EIA		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. **Example:** ●: Hole to be punched  $(0 \times 2^7)+(1 \times 2^6)+(0 \times 2^5)+(0 \times 2^4)+(0 \times 2^3)+(1 \times 2^2)+(1 \times 2^1)+(0 \times 2^0)=\underline{70}$ MPL073
	Program type	M, E	
	Conditions	At I/O startup	
	Unit	—	
	Setting range	0 to 255	
TAP15	"(" code for paper tape reader/puncher for EIA		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. **Example:** ●: Hole to be punched  $(0 \times 2^7)+(0 \times 2^6)+(0 \times 2^5)+(1 \times 2^4)+(1 \times 2^3)+(0 \times 2^2)+(1 \times 2^1)+(0 \times 2^0)=\underline{26}$ MPL074
	Program type	M, E	
	Conditions	At I/O startup	
	Unit	—	
	Setting range	0 to 255	
TAP16	")" code for paper tape reader/puncher for EIA		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. **Example:** ●: Hole to be punched  $(0 \times 2^7)+(1 \times 2^6)+(0 \times 2^5)+(0 \times 2^4)+(1 \times 2^3)+(0 \times 2^2)+(1 \times 2^1)+(0 \times 2^0)=\underline{74}$ MPL075
	Program type	M, E	
	Conditions	At I/O startup	
	Unit	—	
	Setting range	0 to 255	

Classification	DATA I/O	Display title	TAPE

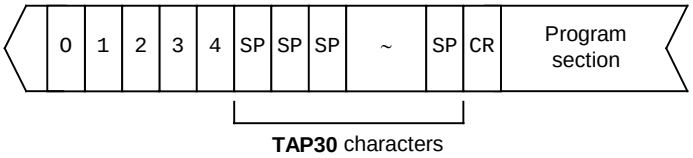
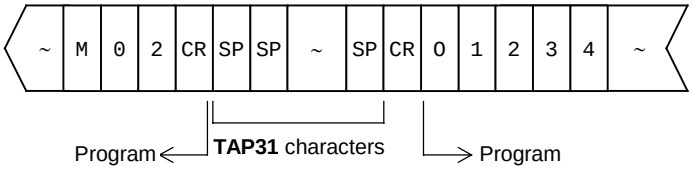
Address	Meaning		Description
TAP17 to TAP24	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
TAP25	Paper tape puncher parity-V check		7 6 5 4 3 2 1 0 0: No parity-V check during paper tape reading 1: Parity-V check during paper tape reading
	Program type	M, E	
	Conditions	At I/O startup	
	Unit	Bit	
	Setting range	Binary, eight digits	
TAP26	Bit parameter related to paper tape reader/puncher		7 6 5 4 3 2 1 0 Data transfer of a paper tape program which has been punched by M2 0: Tape loading impossible 1: Tape loading possible Types of data to be punched onto paper tape during all punching 0: All programs and various NC data 1: All programs only Input/output of the program name on punching/reading of a paper tape 0: No 1: Yes Number of digits of work No. output 0: 8 digits 1: 4 digits Input/output of material data during punching/reading 0: Output of ASCII data in hexadecimal notation 1: Output in characters
	Program type	E	
	Conditions	At I/O startup	
	Unit	Bit	
	Setting range	Binary, eight digits	

Classification	DATA I/O

Display title	TAPE

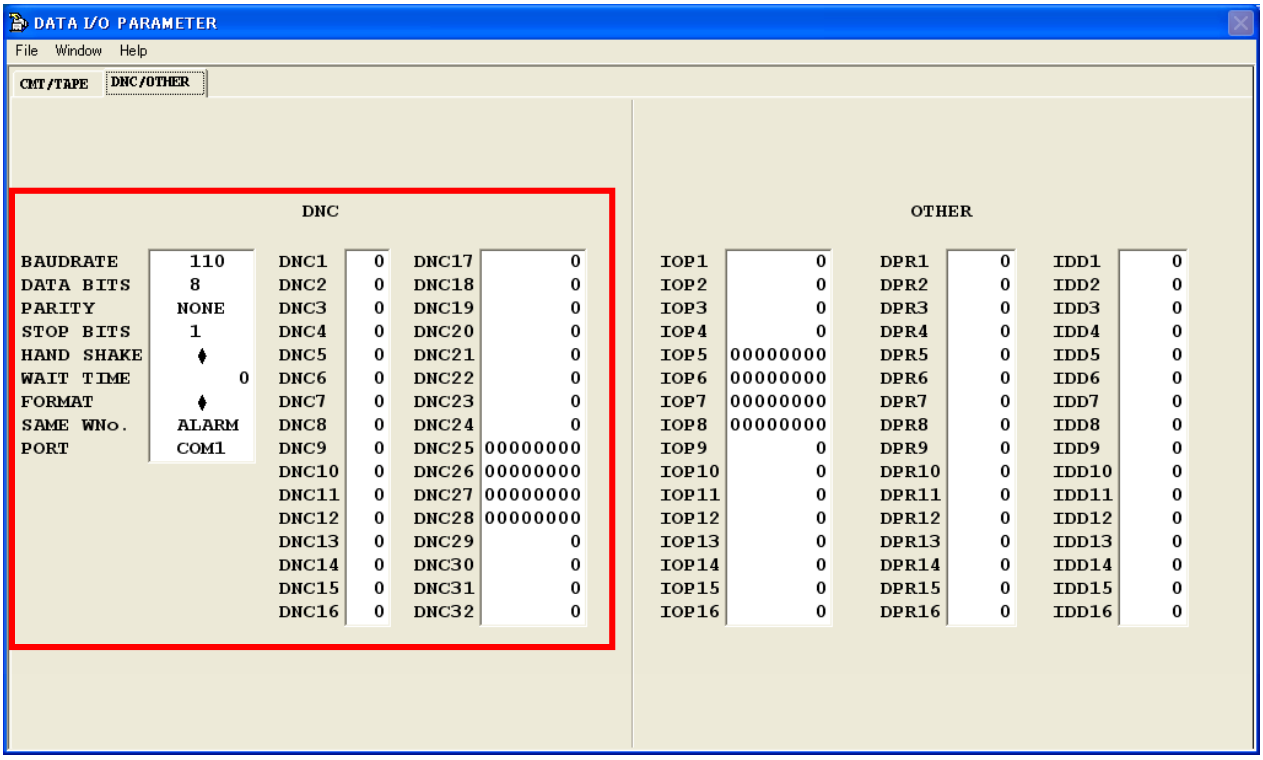
Address	Meaning		Description
TAP27	Bit parameter related to program end code (M) for paper tape reader		
	Program type	E	
	Conditions	At I/O startup	
	Unit	Bit	
	Setting range	Binary, eight digits	
TAP28	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
TAP29	Number of characters in feed section for paper tape puncher		The number of characters in NULL (feed) that are to be punched at the beginning and end of paper tape **Example:**  MPL078
	Program type	E	
	Conditions	At I/O startup	
	Unit	1 character	
	Setting range	0 to 65535	

Classification	DATA I/O	Display title	TAPE

Address	Meaning		Description
TAP30	Number of characters in the space between O-number and program for paper tape puncher		The total number of space-characters that are punched out between O-number and program section.  MPL079
	Program type	E	
	Conditions	At I/O startup	
	Unit	1 character	
	Setting range	0 to 65535	
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	1 character	
	Setting range	0 to 65535	
TAP32	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

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 110 4800 300 9600 1200 19200 2400
Program type	M, E	
Conditions	At I/O startup	
Unit	—	
Setting range	110 to 19200	

Classification	DATA I/O	Display title	DNC

Meaning		Description
DATA BITS		Number of data bits (parameter for RS-232C interface initialization) Set values 7 8
Program type	E	
Conditions	At I/O startup	
Unit	—	
Setting range	7, 8	
PARITY		Parity check (parameter for RS-232C interface initialization) Set values 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 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 sec.	
Setting range	0 to 65535	

Classification	DATA I/O

Display title	DNC

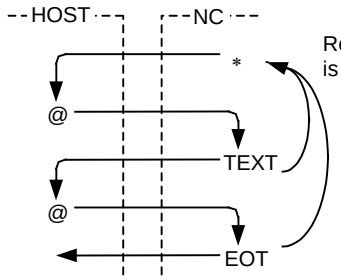
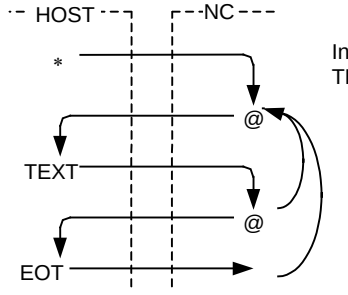
Meaning		Description
SAME WNo.		Type of processing to be executed if the machining program of an existing work number is to be loaded Set values: ALARM — Issues an alarm if the received work number already exists. Set values: LOAD — Overrides the program if the received work number already exists.
Program type	E	
Conditions	At I/O startup	
Unit	—	
Setting range	—	
PORT		DNC port selection Set values: COM1 — CF22 serial ch3 Set values: COM2 — CF22 serial ch4 Set values: COM3 — Invalid Set values: COM4 — Invalid
Program type	E	
Conditions	At I/O startup	
Unit	—	
Setting range	—	

Classification	DATA I/O

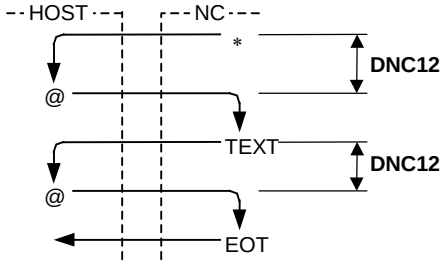
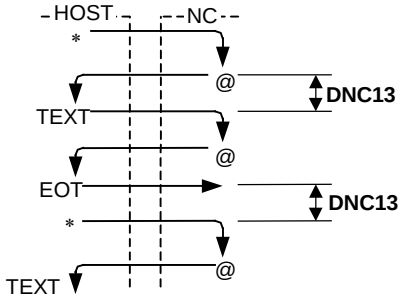
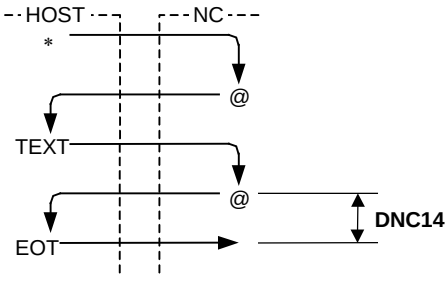
Display title	DNC

Address	Meaning		Description
DNC1	Type of terminator		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	
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	
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	
DNC4	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

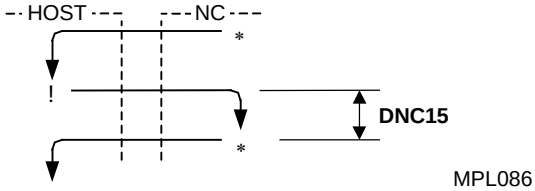
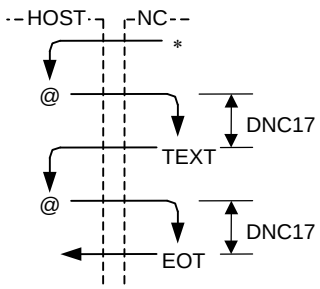
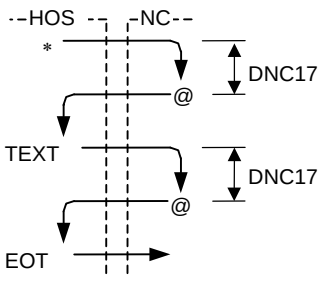
Classification	DATA I/O	Display title	DNC

Address	Meaning		Description
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. Set values: 0 = No assignment, 1 = Assignment Hole-punching pattern of DC3 code: 0 = ○ ○ ○ ● • ○ ○ ○ ● ●; 1 = ● ○ ○ ● • ○ ○ ○ ● ●
	Program type	E	
	Conditions	At I/O startup	
	Unit	—	
	Setting range	0, 1	
DNC6 to **DNC8**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
DNC9	Number of NC transmission retries during DNC file transfer		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.  MPL081
	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		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.  MPL082
	Program type	M, E	
	Conditions	At I/O startup	
	Unit	Number of times	
	Setting range	0 to 255	

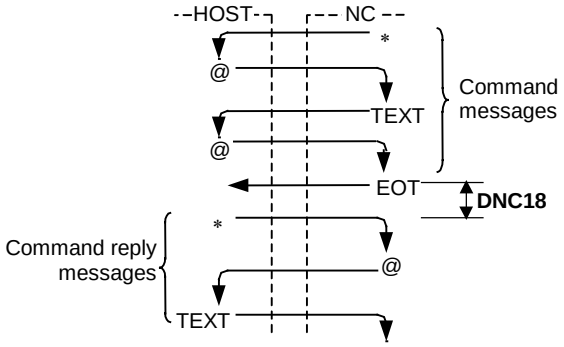
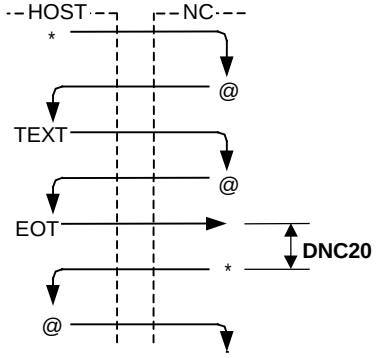
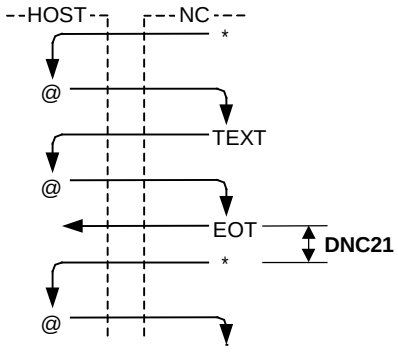
Classification	DATA I/O	Display title	DNC

Address	Meaning		Description
DNC11	Number of NC transmission/reception retries during DNC command message transfer		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**.
	Program type	M, E	
	Conditions	At I/O startup	
	Unit	Number of times	
	Setting range	0 to 255	
DNC12	@ waiting time during DNC transmission		The NC waiting time from transmission of ∗ or TEXT to reception of @ from the host system.  MPL083
	Program type	M, E	
	Conditions	At I/O startup	
	Unit	0.1 sec.	
	Setting range	0 to 255	(⇨ **DNC9**)
DNC13	"∗", TEXT waiting time during DNC transmission		The NC waiting time from transmission of @ or reception of EOT to reception of ∗ or TEXT from the host system.  MPL084
	Program type	M, E	
	Conditions	At I/O startup	
	Unit	0.1 sec.	
	Setting range	0 to 255	(⇨ **DNC10**)
DNC14	EOT waiting time during DNC transmission		The NC waiting time from transmission of @ to reception of EOT from the host system.  MPL085
	Program type	M, E	
	Conditions	At I/O startup	
	Unit	0.1 sec.	
	Setting range	0 to 255	(⇨ **DNC10**)

Classification	DATA I/O	Display title	DNC

Address	Meaning		Description
DNC15	NC stop time after reception of !		The NC stop time from reception of ! from the host system to transmission of ∗.  MPL086
	Program type	M, E	Code ∗ is transmitted to the host system if the time that has been set with **DNC15** elapses following reception of !.
	Conditions	At I/O startup	
	Unit	0.1 sec.	
	Setting range	0 to 255	
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 sec.	
	Setting range	0 to 255	
DNC17	NC stop time from reception		[For NC transmission] The NC stop time from reception of @ from the host system to transmission of EOT or TEXT  [For NC reception] The NC stop time from reception of ∗ or TEXT from the host system to transmission of @  MPL087
	Program type	M, E	
	Conditions	At I/O startup	
	Unit	0.01 sec.	
	Setting range	0 to 255	

Classification	DATA I/O	Display title	DNC

Address	Meaning		Description
DNC18	DNC command reply message waiting time		The NC waiting time from transmission of command message EOT to reception of command reply message ∗ from the host system.  MPL088
	Program type	M, E	
	Conditions	At I/O startup	
	Unit	0.1 sec.	
	Setting range	0 to 255	
DNC19	DNC machine number		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
	Program type	M, E	
	Conditions	At I/O startup	
	Unit	—	
	Setting range	0 to 255	
DNC20	NC transmission stop time of DNC (from reception to transmission)		The NC stop time from reception of EOT from the host system to transmission of ∗ of the next message  MPL089
	Program type	M, E	
	Conditions	At I/O startup	
	Unit	0.01 sec.	
	Setting range	0 to 255	
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 sec.	
	Setting range	0 to 255	

Classification	DATA I/O

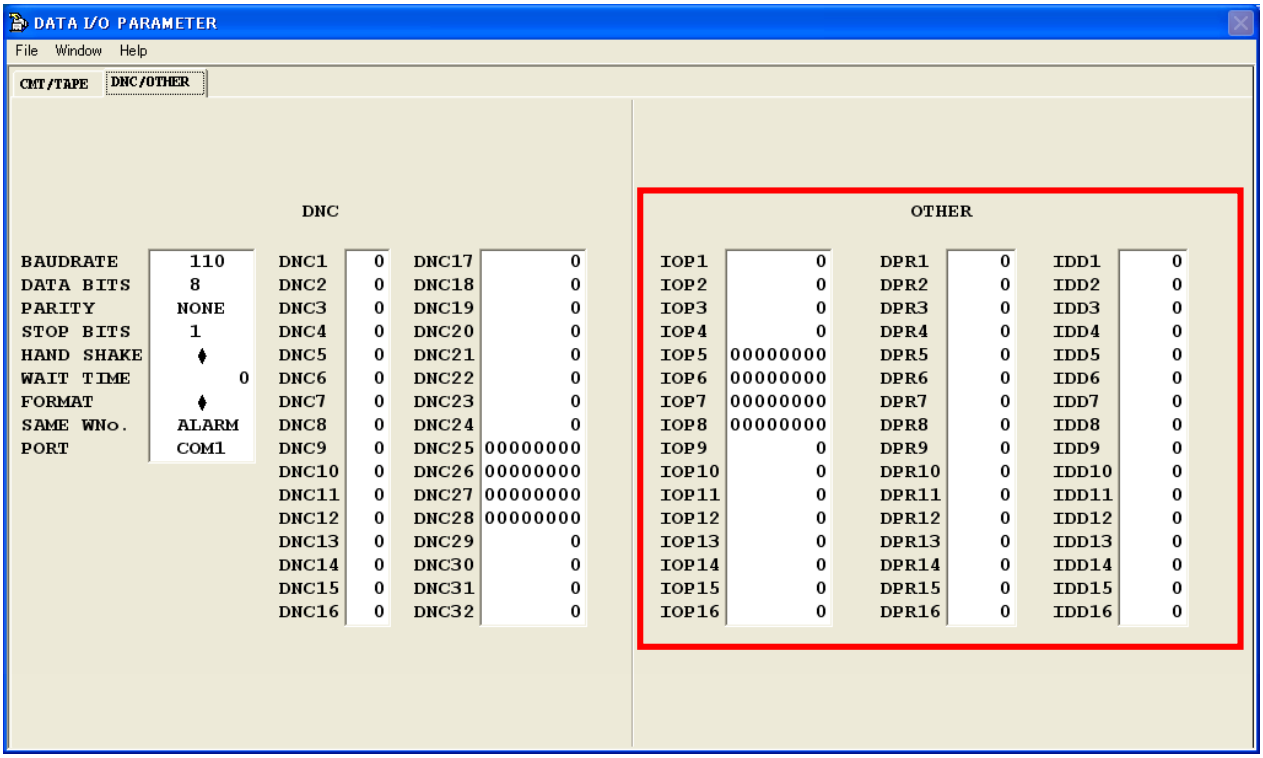
Display title	DNC

Address	Meaning		Description
DNC22 to **DNC24**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
DNC25	—		7 6 5 4 3 2 1 0 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	—		7 6 5 4 3 2 1 0 (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
DNC27 **DNC28**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
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 **DNC32**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

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



D740H0004E

Classification	DATA I/O	Display title	OTHER

Address	Meaning		Description
IOP1 to **IOP4**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
IOP5			7 6 5 4 3 2 1 0 Valid for loading a program(s) 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. 0: Alarm without overwriting 1: Overwriting
	Program type	M, E	
	Conditions	At I/O startup	
	Unit	Bit	
	Setting range	Binary, eight digits	

Classification	DATA I/O	Display title	OTHER

<table>
<tr><th>Address</th><th colspan="2">Meaning</th><th>Description</th></tr>
<tr><td rowspan="5">IOP6</td><td colspan="2">—</td><td>Invalid</td></tr>
<tr><td>Program type</td><td>—</td><td rowspan="4"></td></tr>
<tr><td>Conditions</td><td>—</td></tr>
<tr><td>Unit</td><td>—</td></tr>
<tr><td>Setting range</td><td>—</td></tr>
<tr><td rowspan="5">IOP7</td><td colspan="2">Data entry for communication with the magazine-side display unit</td><td rowspan="5">7 6 5 4 3 2 1 0

Specify the serial port number of the NC unit that is to be used for communication with the magazine-side display unit.
<table><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>
Fixed (0)

Specify the type of ID amplifier.
<table><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>
Specify whether the timing of the R-register is to be made valid during tool removal or mounting.
<table><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></td></tr>
<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>
<tr><td rowspan="5">IOP8</td><td colspan="2">—</td><td>Invalid</td></tr>
<tr><td>Program type</td><td>—</td><td rowspan="4"></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>

Classification	DATA I/O	Display title	OTHER

<table>
<tr><th>Address</th><th colspan="2">Meaning</th><th>Description</th></tr>
<tr><td rowspan="5">IOP9</td><td colspan="2">Number of pitch error axes during text output</td><td rowspan="5">Specify the number of pitch error axes during machine parameter output using the text input/output functions.
Note:
If 0 is entered, machine parameter output will be valid for all axes (14 axes).</td></tr>
<tr><td>Program type</td><td>M, E</td></tr>
<tr><td>Conditions</td><td>At I/O startup</td></tr>
<tr><td>Unit</td><td>Number of axes</td></tr>
<tr><td>Setting range</td><td>0 to 13</td></tr>
<tr><td rowspan="5">IOP10
to
IOP16</td><td colspan="2">—</td><td rowspan="5">Invalid</td></tr>
<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>
<tr><td rowspan="5">DPR1</td><td colspan="2">Baud rate
<table><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></td><td rowspan="5">Baud rate for RS-232C interface
<table><tr><th>Set values</th><th>Baud rate</th><th>Set values</th><th>Baud rate</th></tr><tr><td>0</td><td>110</td><td>4</td><td>4800</td></tr><tr><td>1</td><td>300</td><td>5</td><td>9600</td></tr><tr><td>2</td><td>1200</td><td>6</td><td>19200</td></tr><tr><td>3</td><td>2400</td><td></td><td></td></tr></table></td></tr>
<tr><td>Program type</td><td>M, E</td></tr>
<tr><td>Conditions</td><td>At I/O startup</td></tr>
<tr><td>Unit</td><td>—</td></tr>
<tr><td>Setting range</td><td>0 to 6</td></tr>
<tr><td rowspan="5">DPR2</td><td colspan="2">Stop bit
<table><tr><td>1</td></tr><tr><td>1.5</td></tr><tr><td>2</td></tr></table></td><td rowspan="5">Number of stop bits (parameter for RS-232C interface initialization)
<table><tr><th>Set values</th><th>Stop bit</th></tr><tr><td>0</td><td>1</td></tr><tr><td>1</td><td>1.5</td></tr><tr><td>2</td><td>2</td></tr></table></td></tr>
<tr><td>Program type</td><td>M, E</td></tr>
<tr><td>Conditions</td><td>At I/O startup</td></tr>
<tr><td>Unit</td><td>—</td></tr>
<tr><td>Setting range</td><td>0 to 2</td></tr>
</table>

Classification	DATA I/O	Display title	OTHER

<table>
<tr><th>Address</th><th colspan="2">Meaning</th><th>Description</th></tr>
<tr><td rowspan="5">DPR3</td><td colspan="2">—</td><td>Invalid</td></tr>
<tr><td>Program type</td><td>—</td><td rowspan="4"></td></tr>
<tr><td>Conditions</td><td>—</td></tr>
<tr><td>Unit</td><td>—</td></tr>
<tr><td>Setting range</td><td>—</td></tr>
<tr><td rowspan="5">DPR4</td><td colspan="2">Data bit</td><td>Number of data bits (parameter for RS-232C interface initialization)

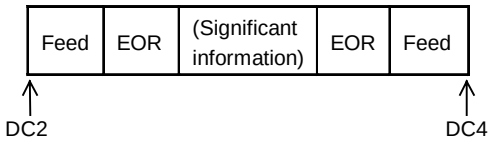
Set values
<table><tr><th>Set values</th><th>Data bit</th></tr><tr><td>0</td><td>8</td></tr><tr><td>1</td><td>7</td></tr></table></td></tr>
<tr><td>Program type</td><td>M, E</td><td rowspan="4"></td></tr>
<tr><td>Conditions</td><td>At I/O startup</td></tr>
<tr><td>Unit</td><td>—</td></tr>
<tr><td>Setting range</td><td>0, 1</td></tr>
<tr><td rowspan="5">DPR5
to
DPR7</td><td colspan="2">—</td><td>Invalid</td></tr>
<tr><td>Program type</td><td>—</td><td rowspan="4"></td></tr>
<tr><td>Conditions</td><td>—</td></tr>
<tr><td>Unit</td><td>—</td></tr>
<tr><td>Setting range</td><td>—</td></tr>
<tr><td rowspan="5">DPR8</td><td colspan="2">ISO code CR output and the output file size</td><td>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).
0: No “CR” insertion
1: “CR” insertion
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).
Maximum permissible file size: Entered value × 100K bytes
However, this file size is 100K bytes if the entered value is 0.</td></tr>
<tr><td>Program type</td><td>M, E</td></tr>
<tr><td>Conditions</td><td>At I/O startup</td></tr>
<tr><td>Unit</td><td>—</td></tr>
<tr><td>Setting range</td><td>0 to 255</td></tr>
</table>

Classification	DATA I/O	Display title	OTHER

<table>
<tr><th>Address</th><th colspan="2">Meaning</th><th>Description</th></tr>
<tr><td rowspan="5">DPR9</td><td colspan="2">Method of handshaking</td><td rowspan="5">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><tr><th>Set values</th><th>Method</th><th>Description</th></tr><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></table></td></tr>
<tr><td>Program type</td><td>M, E</td></tr>
<tr><td>Conditions</td><td>At I/O startup</td></tr>
<tr><td>Unit</td><td>—</td></tr>
<tr><td>Setting range</td><td>0 to 2</td></tr>
<tr><td rowspan="5">DPR10</td><td colspan="2">DC code parity</td><td rowspan="5">This parameter is used to specify whether or not a parity bit is to be assigned to the DC code to be output.

<table><tr><th>Set values</th><th>Parity</th><th colspan="10">Hole-punching pattern of DC3 code</th></tr><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></table>
Note:
This parameter is valid only when the handshaking method is set to DC control (DPR9 is set to 1).</td></tr>
<tr><td>Program type</td><td>M, E</td></tr>
<tr><td>Conditions</td><td>At I/O startup</td></tr>
<tr><td>Unit</td><td>—</td></tr>
<tr><td>Setting range</td><td>0, 1</td></tr>
<tr><td rowspan="5">DPR11</td><td colspan="2">Feed section DC code output</td><td rowspan="5">Select whether or not DC2 and DC4 codes are to be output to the feed sections.

Example:
 
<table><tr><th>Set values</th><th>Description</th></tr><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></table>
Note:
This parameter is valid only when the handshaking method is set to DC control (DPR9 is set to 1).</td></tr>
<tr><td>Program type</td><td>M, E</td></tr>
<tr><td>Conditions</td><td>At I/O startup</td></tr>
<tr><td>Unit</td><td>—</td></tr>
<tr><td>Setting range</td><td>0 to 3</td></tr>
</table>

Classification	DATA I/O	Display title	OTHER

<table>
<tr><th>Address</th><th colspan="2">Meaning</th><th>Description</th></tr>
<tr><td rowspan="5">DPR12</td><td colspan="2">Waiting time</td><td rowspan="5">The waiting time for replies from the connected device during inputting or outputting.
An alarm occurs if this time elapses following the final reply.</td></tr>
<tr><td>Program type</td><td>M, E</td></tr>
<tr><td>Conditions</td><td>At I/O startup</td></tr>
<tr><td>Unit</td><td>0.1 sec.</td></tr>
<tr><td>Setting range</td><td>0 to 65535</td></tr>
<tr><td rowspan="5">DPR13</td><td colspan="2">Output format</td><td rowspan="5">Selection of output code
<table><tr><th>Set values</th><th>Format</th><th>Description</th></tr><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></table></td></tr>
<tr><td>Program type</td><td>M, E</td></tr>
<tr><td>Conditions</td><td>At I/O startup</td></tr>
<tr><td>Unit</td><td>—</td></tr>
<tr><td>Setting range</td><td>0, 1</td></tr>
<tr><td rowspan="5">DPR14</td><td colspan="2">Selection of an output destination port</td><td rowspan="5">Port selection
<table><tr><th>Set value</th><th>Port</th><th>Description</th></tr><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></table><table><tr><th>Set value</th><th>Output destination</th></tr><tr><td>4</td><td>Hard disk (C:\MC_sdg\Print\)</td></tr></table></td></tr>
<tr><td>Program type</td><td>M, E</td></tr>
<tr><td>Conditions</td><td>At I/O startup</td></tr>
<tr><td>Unit</td><td>—</td></tr>
<tr><td>Setting range</td><td>0, 1, 4</td></tr>
<tr><td rowspan="5">DPR15</td><td colspan="2">Number of characters or the number of lines in feed section</td><td rowspan="5">This parameter denotes the number of characters in NULL (feed) when the output destination is a serial port (DPR14 = 0 or 1).

Example:
DPR15 characters DPR15 characters
<table><tr><td>Feed</td><td>EOR</td><td>(Significant information)</td><td>EOR</td><td>Feed</td></tr></table>
This parameter denotes the number of lines fed when the output destination is a hard disk (DPR14 = 4).</td></tr>
<tr><td>Program type</td><td>M, E</td></tr>
<tr><td>Conditions</td><td>At I/O startup</td></tr>
<tr><td>Unit</td><td>1 character</td></tr>
<tr><td>Setting range</td><td>0 to 65535</td></tr>
</table>

Classification	DATA I/O

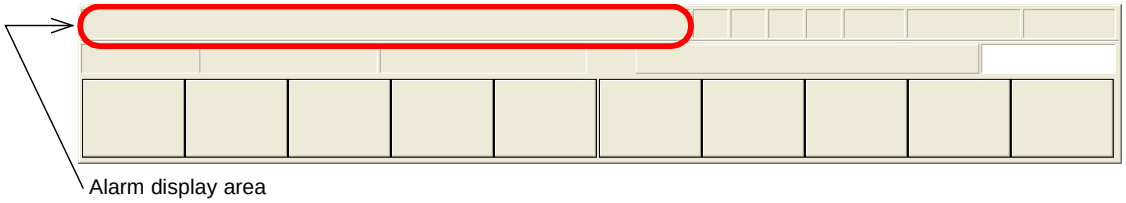
Display title	OTHER

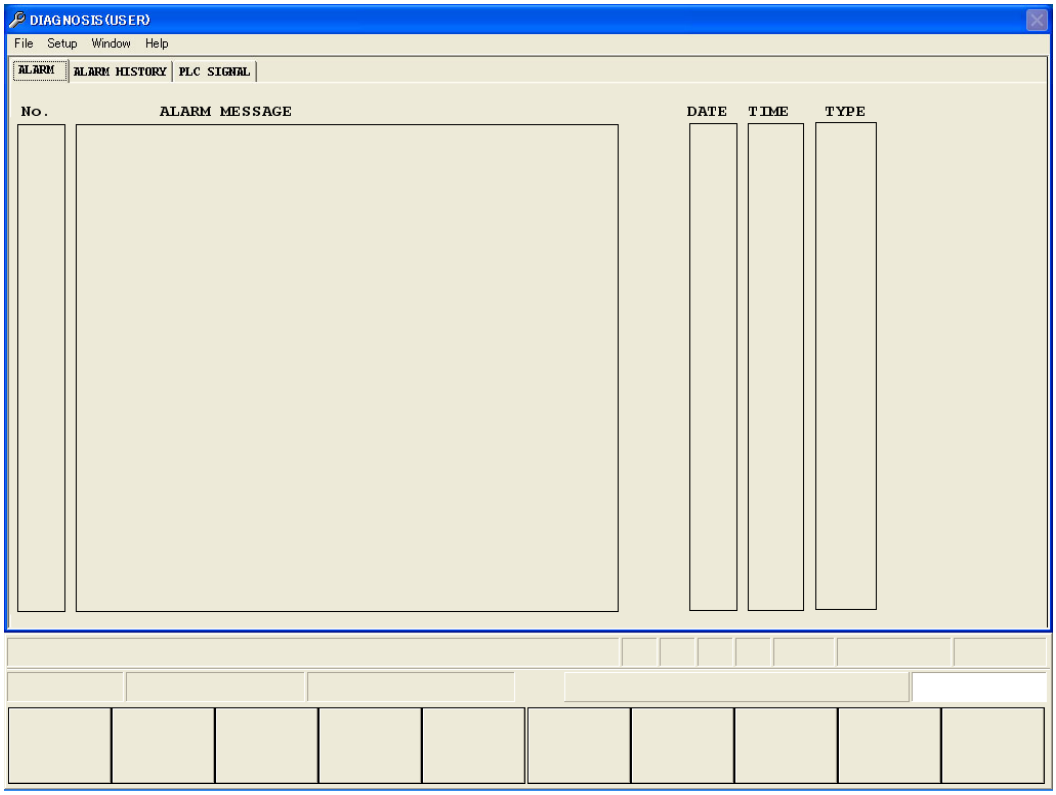
Address	Meaning		Description
DPR16	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
IDD1 to **IDD16**	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

- NOTE -

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.

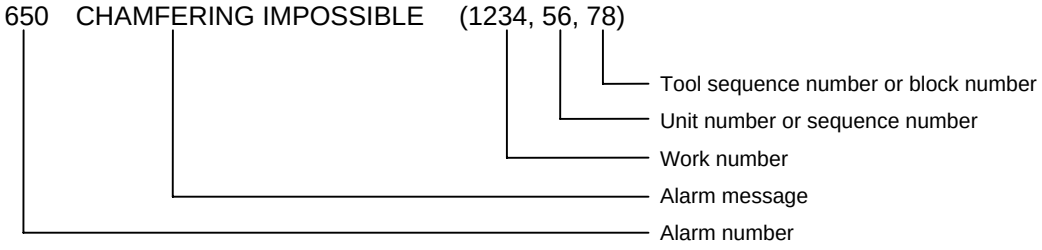
3-1-1 Alarm display

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 either red or blue.

Display color	Alarm elimination
Red	Press the reset key.
Blue	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) has a different format.

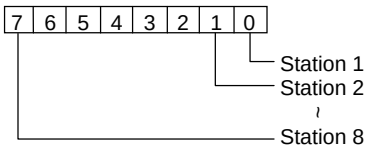
3-2-2 No. 1 - No. 99, No. 1000 - No. 1099 (System/Drive error)

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, require Mazak Technical Center or Technology Center to replace the defective hardware or cables.				
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.				
9	ABSOLUTE POSITION MALFUNCTION (Alarm No., Axis,)	E	H	M	Red
Cause	The absolute position detection system has lost absolute position data.				
Action	Require Mazak Technical Center or Technology Center to re-set the absolute position data.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
10	DETECTOR MALFUNCTION (Alarm No., Axis,)	E	H	M	Red
Cause	The absolute position detection system has detected its detector error(s).				
Action	Require Mazak Technical Center or Technology Center to replace the encoder or bettery.				
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	Require Mazak Technical Center or Technology Center to re-set the absolute position data.				
12	ABSOLUTE POSITION WARNING (Alarm No., Axis,)	E	H	M	Red
Cause	The absolute position detection system has detected abnormal data.				
Action	The battery is running down. Require Mazak Technical Center or Technology Center to replace the battery.				
13	PRE-PROCESSOR MALFUNCTION (, ,)	E	H	M	Red
Cause	The software is not correctly working.				
Action	Require Mazak Technical Center or Technology Center to re-write the NC system data.				
	(, ,)				
Cause					
Action					
21	SYSTEM ERROR (Alarm No., Sub No.,)	E	H	M	Red
Cause	Trouble has occurred during communication with the amplifiers or during communication data processing.				
Action	Contact Mazak Technical Center or Technology Center with the information of the numbers (Alarm No. and Sub No.) displayed in parentheses.				
22	AMPLIFIER NOT EQUIPPED (Axis, ,)	E	H	M	Red
Cause	Amplifier power is not yet turned on. Or no signals are transferred yet. There is a discrepancy between the channel and rotary switch Nos. of the amplifier and the parameters concerned.				
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. Check the settings of parameters **N17**, **N18**, **SA43**, and **SA44** for incorrectness.				
23	—— (, ,)				
Cause	——				
Action	——				
24	—— (, ,)				
Cause	——				
Action	——				

<table>
<tr><th>No.</th><th>Message</th><th>Type of error</th><th>Stopped status</th><th>Clearing procedure</th><th>Display</th></tr>
<tr><td>25</td><td>SAFE OPERATION SYSTEM ALARM (Alarm No., Axis/Door,)</td><td>E</td><td>H</td><td>N</td><td>Red</td></tr>
<tr><td>Cause</td><td colspan="5">There occurred an alarm in the safe operation system.
<table>
<tr><th>Alarm No.</th><th>Cause</th></tr>
<tr><td>1</td><td>Discrepancy between parameters.
The parameters stored in the NC memory for the safe operation system differ from those which are sent to the drive units.</td></tr>
<tr><td>2</td><td>Abnormal speed command.
In the mode of safety-speed operation a command was given for a speed exceeding the speed limit that is specified in the parameter concerned.</td></tr>
<tr><td>3</td><td>Abnormal position feedback.
In the mode of safety-speed operation an excessive error was detected between the position of the command given to the servo drive unit and the feedback position received from there.</td></tr>
<tr><td>4</td><td>Abnormal speed feedback.
In the mode of safety-speed operation the engine speed exceeded the safety limit specified in the parameter concerned.</td></tr>
<tr><td>5</td><td>Discrepancy between input signals of door state.
A difference was detected between the input signal of the door state on the NC side and that on the drive unit side. Probable causes are as follows:
- Break in the cables,
- Defect in the door switch,
- Defect in the NC or servo drive unit.</td></tr>
<tr><td>6</td><td>Abnormal input signal of door open state in the normal mode.
Door open state was detected in the normal mode of operation.
In addition to the same probable causes as for item 5 above, the user PLC might be defective.</td></tr>
<tr><td>7</td><td>Discrepancy between parameters for speed monitoring.
During start-up of the mode of safety-speed operation a discrepancy was detected between two parameters concerned.</td></tr>
</table></td></tr>
<tr><td>Action</td><td colspan="5">Contact Mazak Technical Center or Technology Center with the information of the first item (Alarm No.) displayed in parentheses.</td></tr>
<tr><td>26</td><td>SPINDLE SAFE OPER. SYSTEM ALARM (Alarm No., Axis,)</td><td>E</td><td>H</td><td>M</td><td>Red</td></tr>
<tr><td>Cause</td><td colspan="5">Trouble has occurred for the spindle in the mode of safety-speed operation.
See the description of No. 25 above for the details of probable causes.</td></tr>
<tr><td>Action</td><td colspan="5">Contact Mazak Technical Center or Technology Center with the information of the first item (Alarm No.) displayed in parentheses.</td></tr>
<tr><td></td><td>(, ,)</td><td></td><td></td><td></td><td></td></tr>
<tr><td>Cause</td><td colspan="5"></td></tr>
<tr><td>Action</td><td colspan="5"></td></tr>
<tr><td>31</td><td>SERVO MALFUNCTION 1 (, ,)</td><td>C</td><td>H</td><td>M</td><td>Red</td></tr>
<tr><td>Cause</td><td colspan="5">The servo (power-off level) is abnormal.</td></tr>
<tr><td>Action</td><td colspan="5">Require Mazak Technical Center or Technology Center to replace the servo amplifier.</td></tr>
</table>

No.	Message	Type of error	Stopped status	Clearing procedure	Display
32	SERVO PARAMETER MALFUNCTION (Alarm No., Axis,)	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	Require Mazak Technical Center or Technology Center to re-set the servo parameters.				
33	SERVO MALFUNCTION 2 (Alarm No., Axis,)	C	H	O	Red
Cause	The servo (NC reset level) is abnormal.				
Action	Contact Mazak Technical Center or Technology Center with the information of the first item (Alarm No.) displayed in parentheses.				
34	SERVO MALFUNCTION 3 (Alarm No., Axis,)	C	H	M	Red
Cause	The servo (amplifier power-off level) is abnormal.				
Action	Contact Mazak Technical Center or Technology Center.				
35	OVERLOAD (Alarm No., Axis,)	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 other measures other than the above should be taken, contact Mazak Technical Center or Technology Center with the information of the first item (Alarm No.) displayed in parentheses.				
	(, ,)				
Cause					
Action					
41	SPINDLE MALFUNCTION 1 (Alarm No., Axis,)	C	H	M	Red
Cause	The spindle (power-off level) is abnormal.				
Action	Contact Mazak Technical Center or Technology Center with the information of the first item (Alarm No.) displayed in parentheses.				
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	Require Mazak Technical Center or Technology Center to re-set the spindle parameters.				
43	SPINDLE MALFUNCTION 2 (Alarm No., Axis,)	C	H	O	Red
Cause	The spindle (NC reset level) is abnormal.				
Action	Contact Mazak Technical Center or Technology Center with the information of the first item (Alarm No.) displayed in parentheses.				
44	SPINDLE MALFUNCTION 3 (Alarm No., Axis,)	C	H	M	Red
Cause	The spindle (amplifier power-off level) is abnormal.				
Action	Contact Mazak Technical Center or Technology Center with the information of the first item (Alarm No.) displayed in parentheses.				

<table>
<tr><th>No.</th><th>Message</th><th>Type of error</th><th>Stopped status</th><th>Clearing procedure</th><th>Display</th></tr>
<tr><td></td><td>(, ,)</td><td></td><td></td><td></td><td></td></tr>
<tr><td>Cause</td><td colspan="5"></td></tr>
<tr><td>Action</td><td colspan="5"></td></tr>
<tr><td>51</td><td>E2ROM MALFUNCTION (, ,)</td><td>E</td><td>L</td><td>D</td><td>Blue</td></tr>
<tr><td>Cause</td><td colspan="5">Trouble has occurred during communication with MR-JT-C2.
Parameters cannot be correctly written into the E2ROM.</td></tr>
<tr><td>Action</td><td colspan="5">Require Mazak Technical Center or Technology Center to replace the MR-JT-C2 amplifier.</td></tr>
<tr><td>52</td><td>BATTERY ALARM (, ,)</td><td>E</td><td>L</td><td>D</td><td>Blue</td></tr>
<tr><td>Cause</td><td colspan="5">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.</td></tr>
<tr><td>Action</td><td colspan="5">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.</td></tr>
<tr><td>53</td><td>NC TEMPERATURE WARNING (Note 1, Note 2, Note 3)</td><td>E</td><td>L</td><td>O</td><td>Blue</td></tr>
<tr><td>Cause</td><td colspan="5">The temperature of the control unit or operation board has increased above the required level.
(Note 1: Location of the temperature alarm, Note 2: Type of temperature alarm, Note 3: On-alarm temperature value)</td></tr>
<tr><td>Action</td><td colspan="5">Reduce the temperature by turning off the NC power or by mounting a cooling unit.</td></tr>
<tr><td>54</td><td>DIO5V MALFUNCTION (, ,)</td><td>E</td><td>H</td><td>O</td><td>Blue</td></tr>
<tr><td>Cause</td><td colspan="5">——</td></tr>
<tr><td>Action</td><td colspan="5">Contact Mazak Technical Center or Technology Center.</td></tr>
<tr><td>55</td><td>REMOTE I/O ERROR (Note 1, Note 2,)</td><td>E</td><td>H</td><td>O</td><td>Blue</td></tr>
<tr><td>Cause</td><td colspan="5">——</td></tr>
<tr><td>Action</td><td colspan="5">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~)
 

The above example indicates that a communications error has occurred in station 1 of the No. 2 system.
Note 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.</td></tr>
<tr><td>56</td><td>SYSTEM SOFTWARE CHECKING (, ,)</td><td></td><td>H</td><td></td><td>Red</td></tr>
<tr><td>Cause</td><td colspan="5">The ROMs mounted in the system ROM card are currently being checked for abnormalities.</td></tr>
<tr><td>Action</td><td colspan="5">Wait for a while.
Contact Mazak Technical Center or Technology Center if the alarm is not cleared.</td></tr>
</table>

No.	Message	Type of error	Stopped status	Clearing procedure	Display
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 (Step No., Status, Program No.)	F	H	N	Red/Blue
Cause	Trouble has occurred with the PLC.				
Action	Contact Mazak Technical Center or Technology Center with the information of the second item (Status) displayed in parentheses.				
59	PLC STOPPED (, ,)	F	H	P	
Cause	The PLC has ceased running.				
Action	Contact Mazak Technical Center or Technology Center.				
	(, ,)				
Cause					
Action					
66	PARAMETER MALFUNCTION (, ,)	E	H	O	Blue
Cause	——				
Action	Contact Mazak Technical Center or Technology Center.				
67	ILLEGAL HI-SPEED SYNCTAP AXIS (, ,)	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	ILLEGAL HI-SPEED SYNCTAP SPDL (, ,)	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	——				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
71	ILLEGAL SERVO PARAMETER (Alarm No., Axis,)	C	H	M	Blue
Cause	The parameters that have been transferred from the NC system to the servo amplifier during NC power-on are not correct.				
Action	Contact Mazak Technical Center or Technology Center.				
72	SERVO WARNING (Alarm No., Axis,)	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					
81	ILLEGAL SPINDLE PARAMETER (, ,)	E	H	O	Blue
Cause	The parameters that have been transferred from the NC system to the spindle amplifier during NC power-on are not correct.				
Action	Contact Mazak Technical Center or Technology Center.				
82	SPINDLE WARNING (, ,)	E	H	O	Blue
Cause	The spindle amplifier 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.				
	(, ,)				
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 ZERO (, ,)	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	Red
Cause	Spindle rotation did not start when the spindle rotation start command was issued during automatic operation.				
Action	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. If this alarm should occur seemingly without cause, 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. If this alarm should occur seemingly without cause, 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	Trouble has occurred during pre-interpolation acceleration/deceleration.				
Action	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.				

<table>
<tr><th>No.</th><th>Message</th><th>Type of error</th><th>Stopped status</th><th>Clearing procedure</th><th>Display</th></tr>
<tr><td>144</td><td>ILLEGAL CYCLE START (Alarm No., ,)</td><td>A</td><td>I</td><td>O</td><td>Red</td></tr>
<tr><td>Cause</td><td colspan="5">It was attempted to start automatic operation under the following conditions:
<table>
<tr><th>Alarm No.</th><th>Cause</th></tr>
<tr><td>0 × 101</td><td>During axis movement</td></tr>
<tr><td>0 × 102</td><td>READY lamp OFF.</td></tr>
<tr><td>0 × 103</td><td>Reset state not yet cleared.</td></tr>
<tr><td>0 × 104</td><td>–</td></tr>
<tr><td>0 × 105</td><td>Hardware overtravel not yet cleared.</td></tr>
<tr><td>0 × 106</td><td>Software overtravel not yet cleared.</td></tr>
<tr><td>0 × 107</td><td>–</td></tr>
<tr><td>0 × 108</td><td>Two or more operation modes selected.</td></tr>
<tr><td>0 × 109</td><td>During change in operation modes.</td></tr>
<tr><td>0 × 110</td><td>–</td></tr>
<tr><td>0 × 111</td><td>During search on the tape data.</td></tr>
<tr><td>0 × 112</td><td>–</td></tr>
<tr><td>0 × 113</td><td>Abnormality in temperature.</td></tr>
<tr><td>0 × 114</td><td>During selection of the foreground program.
During editing on the EIA MONITOR display</td></tr>
<tr><td>0 × 115</td><td>A door opened.</td></tr>
<tr><td>0 × 116</td><td>During compensation in tandem control.</td></tr>
<tr><td>0 × 117</td><td>An alarm related to polygonal machining or hob milling not yet cleared.</td></tr>
<tr><td>0 × 118</td><td>An alarm related to safety barrier not yet cleared.</td></tr>
</table></td></tr>
<tr><td>Action</td><td colspan="5">Remove the cause, if possible, to (re)start the automatic operation.
If not, contact Mazak Technical Center or Technology Center with the information of the first item (Alarm No.) displayed in parentheses.</td></tr>
<tr><td>145</td><td>REQUIRE ABSOLUTE RECOVERY (Alarm No., ,)</td><td>C</td><td>K</td><td>P</td><td>Blue</td></tr>
<tr><td>Cause</td><td colspan="5">The absolute position data has become lost.
Trouble has occurred in the absolute position detector.</td></tr>
<tr><td>Action</td><td colspan="5">Require Mazak Technical Center or Technology Center to restore the normal conditions for the zero-point return.</td></tr>
<tr><td>146</td><td>NOT AUTO MODE (, ,)</td><td></td><td></td><td></td><td></td></tr>
<tr><td>Cause</td><td colspan="5">The cycle start button was pressed when the machine was not in the automatic mode.</td></tr>
<tr><td>Action</td><td colspan="5">Change the mode to the automatic and then restart the machine.</td></tr>
<tr><td>147</td><td>C AXIS TURNING ANGLE OVER (, ,)</td><td>A</td><td>K</td><td>P</td><td>Blue</td></tr>
<tr><td>Cause</td><td colspan="5">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.</td></tr>
<tr><td>Action</td><td colspan="5">1. Review the program.
2. Review the setting of parameter K1 (rotational radius of the C-axis).</td></tr>
<tr><td>148</td><td>CHUCK BARRIER (Note, ,)</td><td>A</td><td>K</td><td>O</td><td>Red</td></tr>
<tr><td>Cause</td><td colspan="5">The tool entered in the chuck barrier.
Note: Turning spindle number where alarm has occurred is displayed.</td></tr>
<tr><td>Action</td><td colspan="5">Modify the barrier-related parameters or, for automatic operation, review and correct the program.</td></tr>
</table>

No.	Message	Type of error	Stopped status	Clearing procedure	Display
149	TAILSTOCK BARRIER (, ,)	A	K	O	Red
Cause	The tool entered in the tail barrier.				
Action	Correct the machining program. If the program is correct, review the tool data and the parameters related to the barrier.				
150	WORK PIECE BARRIER (, ,)	A, B	H	O	Red
Cause	Interference is occurring between the workpiece of the No. 1 turning spindle side and the workpiece of the No. 2 turning spindle side.				
Action	Correct the machining program. If the program is correct, review the parameters related to the barrier.				
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	1. The setting of the slave axis name in parameter **M11** is not correct. 2. The name of the master axis that is specified for synchronous control does not correspond with the setting of the axis naming parameter.				
Action	1. Set the name of the slave axis with a small letter. 2. With reference to the parameter, specify the correct axis as the master one for synchronous control.				
153	TRANSFER/PUSH UNFINISHED (, ,)	A	K	O	Red
Cause	1. The axis reached its pressing completion position before a skipping signal was generated. 2. The drooping amount during pressing operation cannot overstep the setting in parameter **K46**.				
Action	1. Check whether the pressing completion position in the program is correct. 2. Modify the **K46** setting as appropriate.				
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 (, ,)	A	K	O	Red
Cause	The barriers of the upper and lower turrets interferes with each other.				
Action	Correct the machining program. If the program is correct, review the parameters related to the barrier.				
156	ILLEGAL MEASURING SENSOR SIGNAL (, ,)	E	L	S	Blue
Cause	The sensor signal for tool measurement turned on during preparation for measurement.				
Action	Start the measurement after the preparation.				
157	MEASURED RESULT MALFUNCTION (, ,)	E	I	P	Red
Cause	All the results of repeated measurements did not fall within the tolerance for compensation.				
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.				

<table>
<tr><th>No.</th><th>Message</th><th>Type of error</th><th>Stopped status</th><th>Clearing procedure</th><th>Display</th></tr>
<tr><td>158</td><td>ILLEGAL COMMAND CROSS MACHINING (, ,)</td><td>A</td><td>K</td><td>Q</td><td>Blue</td></tr>
<tr><td>Cause</td><td colspan="5">It was attempted to manually control an axis currently under the control for cross machining.</td></tr>
<tr><td>Action</td><td colspan="5">The axes under the cross machining control cannot be controlled manually.</td></tr>
<tr><td>159</td><td>CROSS MACHINING IMPOSSIBLE (Alarm No , ,)</td><td>E</td><td>I</td><td>P</td><td>Red</td></tr>
<tr><td>Cause</td><td colspan="5"><table><tr><th>Alarm No.</th><th>Cause</th></tr><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></table></td></tr>
<tr><td>Action</td><td colspan="5">Review and correct the program.</td></tr>
<tr><td>160</td><td>ILLEGAL NUMBER OF CROSS AXIS (, ,)</td><td>E</td><td>I</td><td>P</td><td>Red</td></tr>
<tr><td>Cause</td><td colspan="5">The number of axes in the remote system has become zero as a result of crossing command execution.</td></tr>
<tr><td>Action</td><td colspan="5">Review and correct the program.</td></tr>
<tr><td>161</td><td>SUPERPOSITION CTRL IMPOSSIBLE (WNo., UNo., SNo.)</td><td>E</td><td>I</td><td>P</td><td>Red</td></tr>
<tr><td>Cause</td><td colspan="5">It was attempted to start superposition control with an unavailable axis.
It was attempted to start superposition control under incompatible modal conditions.</td></tr>
<tr><td>Action</td><td colspan="5">Check the parameters related to the type of acceleration & deceleration and to the motion speed during superposition control.</td></tr>
<tr><td>162</td><td>ILLEGAL SUPERPOSITION PARAMETER (WNo., UNo., SNo.)</td><td>E</td><td>I</td><td>P</td><td>Red</td></tr>
<tr><td>Cause</td><td colspan="5">Incorrect parameter settings do not allow the superposition control to be started.</td></tr>
<tr><td>Action</td><td colspan="5">Check the parameters related to the type of acceleration & deceleration and to the motion speed during superposition control.</td></tr>
<tr><td>163</td><td>BARRIER WORK (WNo., UNo., SNo.)</td><td>A</td><td>K</td><td>O</td><td>Red</td></tr>
<tr><td>Cause</td><td colspan="5">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.</td></tr>
<tr><td>Action</td><td colspan="5">Correct the machining program. If the program is correct, review the parameters related to the barrier.</td></tr>
<tr><td>164</td><td>ILLEGAL CMD SAFE OPER SYS ALARM (, ,)</td><td>A</td><td>J</td><td>Q</td><td>Blue</td></tr>
<tr><td>Cause</td><td colspan="5">With the machine under the control for safety speed, a command for the following was given:
Threading, Synchronous tapping, Cross machining, Superposition control.</td></tr>
<tr><td>Action</td><td colspan="5">Close the door to operate the machine.</td></tr>
</table>

No.	Message	Type of error	Stopped status	Clearing procedure	Display
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	In the mode of tool tip point control an unavailable operation (e. g. manual interruption) was attempted.				
Action	Manual interruption cannot be performed in the mode of tool tip point control.				
168	ILLEGAL OPER 5X RADIUS COMP. (, ,)	A	K (O)	O (S)	Blue
Cause	In the mode of tool radius compensation for five-axis machining an unavailable operation (e. g. manual interruption) was attempted.				
Action	Manual interruption cannot be performed in the mode of tool radius compensation for five-axis machining.				
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 SHORTAGE (, ,)	E	J	O	Red
Cause	W-axis thrust has not been obtained.				
Action	Review and correct the program.				
171	W AXIS THRUST SUPPLYING (, ,)	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.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
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	ILLEGAL COMMD TIME CONST. CHANGE (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.				
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.				
	(, ,)				
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.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
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 circumferential 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.				
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.				
	(, ,)				
Cause					
Action					

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 cancel key to clean the alarm.				
1102	INTERFERE (, ,)	A	I	P	Red
Cause	Interference occurs between <Interfering section 1> and <Interfering section 2>.				
Action	Press the reset key to clean the alarm.				
1103	INTERFERE CHECK DATA OVERLOAD (, ,)	B	L	S	Blue
Cause	The Intelligent Safety Shield process was not in time.				
Action	Set a simple workpiece model or a fixture model or a tool model.				
	(, ,)				
Cause					
Action					

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

For alarms related to PLC machine control (No. 200 to No. 399 and No. 1200 to No. 1399) refer to the Operating Manual of the machine.

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.				

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. - An attempt has been made to erase the common unit during editing of the MAZATROL program.				
Action	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	Automatic calculation of circumferential speed and feedrate is not possible. **Note:** The sub-error codes displayed when the **[NAVIGATE]** menu function is selected on the **MACHINING NAVIGATION–PREDICTION** display are listed below. –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 velocity 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 velocity data tables –8: Tool materials mismatch error 2: 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	1. The pitch or other data cannot be automatically set because of incorrectness of the tap nominal diameter in the tapping-unit data. 2. 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	1. Check and correct the tapping-unit data and tapping-tool sequence data of the program. 2. Check and correct the auto-setting text file (Pipescdt. txt).				
420	SAME DATA EXISTS (, ,)	A	L	S	Blue
Cause	An attempt has been made to input the same data as that which has already been registered. 1. Pocket number in the **TOOL LAYOUT** display. 2. Machining-program number (changed) 3. Machining priority number				
Action	Check and correct the data settings.				
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.				

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-ϕ 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).				
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.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
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.				
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.				
Action	Press the reset key and then carry out the restart operation once again.				
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. **Example 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. **Example 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. **Example 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	PRINTER IN OPERATION (, ,)	A	L	S	Blue
Cause	An attempt was made on the **TRACE** display to perform display scaling change, material shape and tool path drawing while the hard copy is being made.				
Action	After finishing the hard copy, carry out the operations.				
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	MEMORY PROTECT (SAMPLING) (, ,)	A	L	S	Blue
Cause	An attempt was made during load sampling to change the axis to be monitored or the sampling time.				
Action	Perform the required change after the current sampling.				
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.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
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.				
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.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
482	—— (, ,)				
Cause	——				
Action	——				
483	SOLID DESCRIP. IMPOSSIBLE UNIT (WNo., UNo.,)	B	I	S	Blue
Cause	Checking the program including the units or conditions (such as the case that angle B is set in the index unit of the initial-point scheme) that do not allow graphics to be drawn in SOLID mode has been attempted in the SOLID mode of the **TOOL PATH CHECK** display.				
Action	Before checking the tool path, cancel the SOLID mode or delete the unit that cannot be drawn in the SOLID mode.				
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 degrees, 90 degrees, nor 180 degrees.				
Action	Execute the tool nose position storage function (TEACH) when the head index angle is either 0 degrees, 90 degrees, or 180 degrees.				
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 store the current position for re-threading (using the **[THR. POS. TEACH]** menu function) without the turning spindle having been rotated through at least one full turn.				
Action	Rotate the turning spindle at least through one full turn beforehand to use the above menu function.				
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.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
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.				
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	An attempt was made to set jaw dimensions without having selected the jaw type (OUT1/IN/OUT2).				
Action	Select the jaw type beforehand.				
494	NO TOOL SELECT (NOT MGZN TYPE) (, ,)	A	L	S	Blue
Cause	An attempt was made to use the MDI tool selection function on the machine with a turret-type tool-post.				
Action	——				
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 operation with the head not in its horizontal or vertical machining position.				
Action	1. Tool tip position storage (tool length measurement) is possible only with the head in its horizontal machining position. 2. MDI-MMS 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.				

No.	Message	Type of error	Stopped status	Clearing procedure	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.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
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.				
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 mode of operation.				
Action	Change the operation mode.				
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.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
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.				
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.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
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.				
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	NOT POCKET FOR LONG BORING BAR (, ,)	A	L	S	Blue
Cause	Assigning the long boring bar to a section not defined as a special pocket for the long boring bar has been attempted during the “Layout Completion” operations on the **VISUAL TOOL MANAGEMENT** display.				
Action	Assign the long boring bar to the special pocket.				
1429	INCORRECT SELECT TOOL (, ,)	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	Tool data				
Action					
1432	NO SELECT TOOL MODEL (, ,)	B	L	S	Blue
Cause	The tool model to be copied/erased/renamed is not selected.				
Action	Select the appropriate tool model before performing the copy/erasure/name change.				
1433	SAME TOOL MODEL EXISTS (, ,)	B	L	S	Blue
Cause	A registered tool model has been specified during the copy/name change.				
Action	Specify an unregistered name.				
	(, ,)				
Cause					
Action					

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

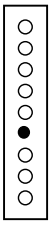
No.	Message	Type of error	Stopped status	Clearing procedure	Display
500	—— (, ,)				
Cause	——				
Action	——				
501	ILLEGAL FORMAT (, ,)	A	L	S	Blue
Cause	An external storage medium that contains data other than MATRIX data has been set.				
Action	Check the contents of the external storage medium for 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.				

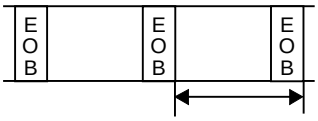
No.	Message	Type of error	Stopped status	Clearing procedure	Display
508	MEMORY CAPACITY EXCEEDED (WNo., ,)	A (G)	L (L)	S (S)	Blue (Blue)
Cause	1. An attempt has been made to load machining program data that exceeds the maximum available area for program registration within the NC unit. 2. In the middle 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	—— (, ,)				
Cause	——				
Action	——				
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	——				
523	—— (, ,)				
Cause	——				
Action	——				

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.				
	(, ,)				
Cause					
Action					
530	CMT MIS-CONNECTED (, ,)	G	L	S	Blue
Cause	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. 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. 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-use disk does not contain a saved machining program (only machining programs can be loaded from M640-use disks).				
Action	The disk that has been registered for M640 use does not contain a registered machining program. Perform checks using the NC unit M640.				
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.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
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	 ← Sprocket 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	The contents of the paper tape or floppy disk cannot be read since they include a parity-V error(s).  The number of sprocket holes in this area must be even.				
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	A machining program in which EOR precedes the end M-code (M02, M30 or M99) or the next O number (work number) was loaded. The end-of-program condition can be changed by varying the settings of the parameter (**TAP27**).				
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	- 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	Check for incorrect cable connections. Or check if the power is turned on.				
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	—— (, ,)				
Cause	——				
Action	——				
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.)				
571	ILLEGAL FORMAT (, ,)	A	L	S	Blue
Cause	Data other than MATRIX use data has been transmitted from the host system. (The format of the transmitted data is not correct.)				
Action	Check the transmitted data for appropriateness to the MATRIX.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
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	- 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	DISK NOT READY (, ,)	A	L	S	Blue
Cause	A disk has not been correctly mounted.				
Action	Correctly mount a disk in the 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	- Check if the program that has been transferred from the host system is for use with MATRIX. - Check if the contents of the file transfer message (header block) are correct.				
584	RECEIVED DATA TYPE INCORRECT (, ,)	A	L	S	Blue
Cause	- 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	- 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	- Check if the DNC cables are correctly connected. - Check if the host system is turned on and ready for data transmission/reception. 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.				
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.)				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
587	PROG. OPERATION NOT ALLOWED (WNo., ,)	A	L	S	Blue
Cause	1. An attempt has been made to transmit a display inhibiting program to the host system. (Program management function) 2. 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	1. Check if the specified work number is for the program of display inhibition. 2. Carry out the transfer operation only after completion of the program editing (or program loading using another I/O unit).				
588	DATA OPERATION NOT ALLOWED (, ,)	A	L	S	Blue
Cause	- 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	The EIA/ISO machining program includes a block that consists of more than 256 characters. (EOB or EOR is not present 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 machining program. (Insert EOB within 256 characters.)				
590	DNC COMMAND IMPOSSIBLE (, ,)	A	L	S	Blue
Cause	The particular status of the NC unit disables execution of the control command that has been requested from the host system. - 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	- 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	- 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**).				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
593	DNC I/O ERROR (, ,)	G	L	S	Blue
Cause	- 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	- 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	- 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	- 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)				
595	FILE TRANSFER ERROR (, ,)	G	L	S	Blue
Cause	- 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	- 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. - 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	- 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	- 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 excess 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 circumferential speed (**C-SP**) is unset or set to “0”.				
Action	Review the machining program and set the desired circumferential 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	- 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	- 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	NO TOOL IN MAGAZINE (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.				
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.				
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.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
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.				
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 { H : M39, L : M38 } 3-gear { H : M39, M : M38, L : M37 } 4-gear { H : M39, MH : M38, ML : M37, L : M36 }				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
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.				
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.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
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	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.				
Action	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.				
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.				
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”.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
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.				
676	ILLEGAL UNIT (WNo., UNo.,)	B	I (L)	O (S)	Red (Blue)
Cause	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. 2. An MMS unit has been set in the program that has table rotational machining designated in its face definition sequence.				
Action	1. Remove all illegal units and sequences from the machining program. 2. MMS measurement is not possible while table rotational machining is effective.				
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.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
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 in 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.				
682	ILLEGAL REPEAT FIGURE (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	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.				
Action	Review and correct the corresponding shape data.				
683	UNDEFINED CORNER (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The value of designated corner rounding or corner chamfering is not appropriate.				
Action	Review the corresponding shape data and set the correct corner rounding or corner chamfering value.				
684	POINT CUTTING PATTERN ERROR (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The point-machining shape definition data is not appropriate.				
Action	Review and correct the corresponding shape data.				
685	SQUARE CANNOT BE DEFINED (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	When the shape pattern is “square”, the input data cannot be used to make shape definitions.				
Action	Review and correct the corresponding shape data.				
686	NO STARTING POINT (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	During input of free-shape data (open-figure data) to the line-machining unit, “?” has been set as the definition of the starting point.				
Action	Review the machining program and set the coordinates of the starting point of the free shape.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
687	NO FINISH POINT (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	During input of free-shape data (open-figure data) to the line-machining unit, “?” has been set as the definition of the ending point.				
Action	Review the machining program and set the coordinates of the ending point of the free shape.				
688	INSUFFICIENT INPUT DATA (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The coordinates of the intersection point cannot be calculated since the free-shape input data in the line- or face-machining unit is incomplete.				
Action	Review the corresponding shape data and set data that is wanting.				
689	INPUT DATA TOO MANY (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	1. The line- or face-machining unit contains too much free-shape input data, and there is contradiction between overlapping data. 2. Too many tool sequences have been set for the line- or face-machining unit.				
Action	1. Review the corresponding shape data and erase either one of the overlapping data sets. 2. Reduce the number of tool sequences.				
690	ILLEGAL RADIUS (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Contradiction exists in the free-shape data that have been set to define arc in the line- or face-machining units.				
Action	Review the corresponding shape data and set correct data.				
691	MOUNT (VALLEY) SHAPE ERROR (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The second figures (inside figures) are not yet defined in the endmilling-mountain (STEP), pocket milling-mountain or pocket milling-valley unit.				
Action	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) (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The number of points which are necessary to define the shapes designated in the line- or face-machining unit exceeds 200.				
Action	Review the machining program, and reduce the number of shapes within one line- or face-machining unit.				
693	NUMBER OF SHAPES TOO LARGE (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Among the line- or face-machining units is one(s) that contains more shapes than allowable within one unit.				
Action	Review the corresponding shape data and check the number of shapes.				
694	FIXED FIGURE DESIGNATED ERROR (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	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.				
Action	Change the fixed shapes to free ones.				
695	POINT INSIDE CIRCLE (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	It is not possible to draw a tangent line from point P1 since it is inside the arc.				
Action	Review the machining program and check the free-shaped data.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
696	ILLEGAL DIRECTION (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	“Left” or “right” is set for the crossing point (P), though “up” or “down” should have been set.				
Action	Review the machining program and check the value of the crossing point (P).				
697	DATUM <P> REQUIRED (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	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.				
Action	Review the machining program and set the crossing point (P).				
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.				

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	——				
Action	——				
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.				
Action	Change the shape compensation clearance (parameter **E13**) to an appropriate value: or use a tool of smaller diameter.				
702	FIGURE DEFINITION ERROR (WNo., UNo., SNo.)	B	K	O	Blue
Cause	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.				
Action	Such contradiction usually results from arithmetic errors. Change the radial depth of cut by some micro, or use a tool of smaller diameter.				
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).				
Action	Change the machining conditions to correct ones.				
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.				
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).				
705	APPROACH POINT ERROR (WNo., UNo., SNo.)	B	K	O	Blue
Cause	The approach point cannot be obtained.				
Action	Reduce the tool diameter, the approach amount (parameters **E1**, **E2**) and/or the overlap amount (parameter **E21**).				
706	ILLEGAL FIGURE DATA (WNo., UNo., SNo.)	B	K	O	Blue
Cause	- 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.				
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.				
707	CHAMFER CUTTER INTERFERENCE (WNo., UNo., SNo.)	B	K	O	Blue
Cause	The chamfering tool interferes with the side wall or bottom.				
Action	Use a tool which does not interfere with the side wall or bottom.				

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

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 peripheral velocity 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**.				

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) Specified by parameter **TC78**. Calculated from the height (HGT) and the number of times (NUMBER) in unit data.				
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	Type of error	Stopped status	Clearing procedure	Display
742	ILLEGAL THREAD HEIGHT (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	In the THREAD unit data, the following condition is not satisfied. (Finishing allowance) ≤ (Thread height) Designated in the unit data. Set using parameter **TC78**.				
Action	Change the thread height or the setting of parameter **TC78**.				
743	INTERSECTION NOT FOUND (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	No automatic calculations of intersection were obtained since there are shape sequence data disparities.				
Action	Correct the shape sequence data.				
744	DATA MISSING (INTERSECTION) (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	No automatic calculations of intersection were obtained since there are lack of data to automatically calculate a point of intersection.				
Action	Correct the unit data or the sequence data.				
745	INTERSECTION CALCULATE IMPOSS. (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	No automatic calculations of intersection were obtained since there are illegal data in the program.				
Action	Correct the unit data or the sequence data.				
746	NO DEPTH OF CUT INFO. (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	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.				
Action	Set data in **DEPTH** or **CUT ANGLE**, or use some other tool.				
747	NO CHIP CUTTING CYCLE OPTION (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	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.)				
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	CUT AREA EXCEEDED (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	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.				
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.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
749	——				
Cause	——				
Action	——				
750	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.				
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	The check surface values are incorrect. 1. For rough-machining 1 or 2: Check surface Z min. ≥ material height 2. For finishing: Check surface Z min. > initial Z 3. Check surface X min. > X max., or Y min. > Y max., or Z min. > Z max.				
Action	Set the check surface values as follows: 1. For rough-machining 1 or 2: Check surface Z min. < material height 2. For finishing: Check surface Z min. ≤ initial Z 3. Check surface X min. ≤ X max., and Y min. ≤ Y max., and Z min. ≤ 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).				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
757	CURVE DEFINITION LARGE (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	1. For rough-machining 2 with designation of workpiece size, the dimensions of the defined figure are larger than those of the workpiece. 2. 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".				
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	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.				
Action	Before creating C-axis machining or turning unit, create a WPC unit with reference to the axis of turning.				
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.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
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.				
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 on a MULTIPLEX machine: 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	For a MULTIPLEX machine: 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	For the execution of a composite program on a MULTIPLEX machine: Machining units are not preceded by a head selection unit.				
Action	Set a HEAD 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 on a MULTIPLEX machine: 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.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
774	HEAD UNIT IN PROGRAM (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	For the execution of an independent program on a MULTIPLEX machine: 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.				
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	RESTART IMPOS. LO-TURRET (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	SINGLE PROCE. IMPOS. LO-TURRET (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.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
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.				
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	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.				
Action	Set the “2 Workpiece” machining units in correct order.				
790	ILLEGAL BALANCE CUT (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Only a single line of tool sequence is created for balanced cutting.				
Action	Add another tool sequence line to use the other turret symmetrically for balanced cutting.				
791	ILLEGAL CUTTING SPEED (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The peripheral speed data set in the tool sequence contains an invalid value.				
Action	Set correct peripheral speed data in the tool sequence.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
792	ILLEGAL 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.				
793	ILLEGAL MILL AXIS RPM (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	If **DRILL** is selected as the turning drilingl 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.				
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. Increase the spindle speed for the turning spindle.				
794	HEAD ANGLE INCORRECT (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	When M Pro system is used in 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 degrees.				
795	—— (, ,)				
Cause	——				
Action	——				
796	NO LO-TURRET MAZATROL OPTION (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 turrret control option is unavailable.				
797	BARRIER CANCEL ON (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The "Cancel Barrier" setting is enabled with the Prohibit Startup when Barrier is Disabled function enabled.				
Action	Disable the "Cancel Barrier" setting.				
798	JAW NO NOT FOUND (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The jaw No. setting does not exist although the Prohibit Startup when Barrier is Disabled function is enabled.				
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	——				
Action	——				
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	The format in which the data has been designated in the program is incorrect.				
Action	Review the program.				
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.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
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).				
817	INCORRECT ARC DATA (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The relationship between the starting and ending points of the arc and the center of the arc is not appropriate.				
Action	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).				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
818	MISSING CENTER (NO DATA) (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	For arc interpolation by R designation, the coordinates of the center of the arc cannot be calculated.				
Action	Correct the value of each address in the program.				
819	HELICAL OPTION NOT FOUND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The helical interpolation command has been issued although such an interpolation function is not provided.				
Action	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 (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The spiral interpolation command (G02.1 or G03.1) has been issued although such an interpolation function is not provided.				
Action	Delete the G02.1 or G03.1 command.				
821	G07 OPTION NOT FOUND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The virtual-axis command (G07) has been issued although there are not virtual-axis specifications.				
Action	Check the specifications, and then change the G07 command.				
822	ILLEGAL MODAL (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	A G-code command has been specified in illegal modal data.				
Action	Review the program.				
823	G17 - G19 COMMAND IN M98 (WNo., NNo., BNo.)	B	I	O	Red (Blue)
Cause	A plane selection command (G17, G18 or G19) has been issued during figure rotation.				
Action	Delete the plane selection command (G17, G18 or G19) from the figure rotation subprogram.				
824	G17 - G19 COMMAND IN G68 (WNo., NNo., BNo.)	E	I	O	Red (Blue)
Cause	A plane selection command (G17, G18 or G19) has been specified in the coordinates rotation command (G68).				
Action	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 (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	A plane selection command (G17, G18 or G19) has been specified during tool diameter offset (G41 or G42).				
Action	Specify the plane selection command after the tool diameter offset command has been canceled by G40.				
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.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
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 circumferential speed command (G96) has been specified although such specifications are not provided.				
Action	Check the specifications and change the constant circumferential speed command (G96) to the speed command (rpm).				
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.				
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).				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
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 (D○○); 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	O	Red
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	1. The total number of sequential calls of subprogram has exceeded eight. 2. Executing a program that includes an “M99” command has been attempted in the direct operation mode of HD, IC memory card or the Ethernet. 3. 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	1. Check the number of subprogram calls, and correct the program so that the number of calls does not exceed eight. 2. Do not include an “M99” command in the main program to be executed in the direct operation mode. 3. 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.				
843	DESIGNATED SNo. NOT FOUND (WNo., NNo., BNo.)	B	K	S	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.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
844	PROGRAM No. NOT FOUND (WNo., NNo., BNo.)	B	K	S	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	K	S	Blue
Cause	A variables number has been designated although variables number (#○○) specifications are not provided.				
Action	Check the specifications.				
846	DESIGNATED NUMBER NOT FOUND (WNo., NNo., BNo.)	B	K	S	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	K	O	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.				
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.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
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	O	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.				
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.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
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.				
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.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
871	VARIABLE NAME EXISTS (WNo., NNo., BNo.)	A	K (L)	P (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.				
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.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
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	I (L)	O (S)	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.				
888	FILE I/O ERROR (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	The machining program file cannot be read.				
Action	Contact Mazak Technical Center or Technology Center.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
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	In the thread cutting cycle command, designation for chamfering is incorrect.				
Action	Set chamfering data that ensures termination of the operation of the tool within the cycle.				
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).				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
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; …				
	(, ,)				
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.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
909	ILLEGAL PITCH IN FIXED CYCLE (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (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	I (L)	O (S)	Red (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	J (L)	P (S)	Red (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	J (L)	P (S)	Red (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	I (L)	O (S)	Red (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	I (L)	O (S)	Red (Blue)
Cause	In the geometric command, the difference in angle between the two straight lines which intersect with each other is less than 1 degree.				
Action	Increase the angle difference in the geometric command.				
916	GEOMETRIC COMMAND NOT ABSOLUTE (WNo., NNo., BNo.)	B	K	O	Red (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	J (L)	P (S)	Red (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	J (L)	P (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	J (L)	P (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	J (L)	P (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	J (L)	P (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	J (L)	P (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. MAZAK precision rapid boring tornado option or 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 G-code of inhibition.				
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. <HV>				
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. <HV>				
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. <HV>				
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 the execution of EIA/ISO program or of MDI, the tool data was found not to include **LENGTH** or **ACT-ϕ** (**NOM-ϕ**) (this, however, applies only if bit 7 of parameter **F84** is set to "1" for the use of MAZATROL tool length and tool diameter data).				
Action	Recheck the tool data and set missing values. (Related parameters: **F84** bit 7, **F92** bit 7, **F93** bit 3)				
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.)	B	I (L)	O (S)	Red (Blue)
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.)	B	I (L)	O (S)	Red (Blue)
Cause	Restarting from the block containing the G06.2 mode data has been attempted.				
Action	Restart from a block not containing G06.2 mode data.				
957	MANUAL INTERRUPT NOT ALLOWED (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
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.)	B	I (L)	O (S)	Red (Blue)
Cause	1. The threshold value for load detection-based auto-pecking is not set to 0 or no such value is set. 2. The parameter is not set appropriately.				
Action	1. Set the appropriate threshold value either in the drill monitoring mode of the **MACHINING-MONITORING** display or on the **TOOL DATA** display. 2. For parameter setting, contact Mazak Technical Center or Technology Center.				
959	WORKPIECE COORDINATE ERROR (, ,)	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.				
960	SUPERPOSIT CTRL ILLEGAL COMMAND (, ,)	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	A preparatory function which is incompatible with the mode of tool diameter offset for five-axis machining has been designated.				
Action	Review the program, and delete the illegal G-code.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
962	CANNOT USE 5X RADIUS COMP. (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	A command of tool diameter offset for five-axis machining has been designated under incompatible modal conditions.				
Action	Review the program, and delete the illegal G-code.				
963	TURRET MIRROR IMAGE N/A (, ,)	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	J (L)	P (S)	Red (Blue)
Cause	The parameter settings on the composition of controlled axes for tool tip point control are not correct.				
Action	Correct the parameter settings concerned.				
971	CANNOT USE TOOL TIP PT CONTROL (WNo., NNo., BNo.)	B	J (L)	P (S)	Red (Blue)
Cause	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	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.				
972	ILLEGAL CMD TOOL TIP PT CTRL (WNo., NNo., BNo.)	B	J (L)	P (S)	Red (Blue)
Cause	An unavailable command (of preparatory or tool function) is given in the mode of tool tip point control.				
Action	Temporarily 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	J (L)	P (S)	Red (Blue)
Cause	An inappropriate command relevant to the vector of tool axis is given in the mode of tool tip point control. 1. 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. 2. The command of the vector of tool axis is not correct (as it will reverse the direction of the tool). 3. 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	1. Modify the program for a tool path through the singular point, or select joint interpolation. 2. Divide the command block concerned in order that the resulting rotation of the tool direction per block will not exceed 180°. 3. Modify the program for a per-block motion angle of 180° or less on the rotary axis of linear type, or select joint interpolation.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
974	TOOL TIP PT CTRL FORMAT ERROR (WNo., NNo., BNo.)	B	J (L)	P (S)	Red (Blue)
Cause	1. 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. 2. The code G49 is given in a block with other commands. 3. A rotary axis is specified in the mode of circular interpolation.				
Action	1. 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. 2. Give the cancellation command G49 in a single-command block. 3. A rotary axis cannot be specified in the mode of circular interpolation.				
975	TOOL TIP PT CTRL N/A (WNo., NNo., BNo.)	B	J (L)	P (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 (, ,)				
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.				
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.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
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.				
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.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
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.				
Action	1. Contact Mazak Technical Center or Technology Center. 2. Refer to the relevant user macroprogram instruction manual to check the alarm.				
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.				
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	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: 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. 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.				
	(, ,)				
Cause					
Action					
2120	RESTART PT SEARCH INTERRUPTED (, ,)	A	L	S	Blue
Cause	The restart positon 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					

No.	Message	Type of error	Stopped status	Clearing procedure	Display
2130	CANNOT DISPLAY WORKPIECE MODEL (, ,)	A	L	S	Blue
Cause	1. Lack of available memory space 2. Arithmetical incapability 3. Entered data imperfections				
Action	1. Divide the program. 2. Confirm parameters. 3. Confirm tool data.				
	(, ,)				
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.				
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**.				

4 M-CODE LIST

For description of M-codes, refer to the Operating Manual of the machine.

- NOTE -