



Changes for the Better

MITSUBISHI CNC

700/70 Series

Instruction Manual



Microsoft® and Windows® are either registered trademarks or trademarks of Microsoft Corporation in the United States and/or other countries.

MELDAS is a registered trademark of Mitsubishi Electric Corporation.
Other company and product names that appear in this manual are trademarks or registered trademarks of the respective companies.

Introduction

This manual is referred to when using the MITSUBISHI CNC 700/70 Series.
This manual explains how to operate, run and set up this NC unit. Read this manual thoroughly before using the NC unit. To safely use this NC unit, thoroughly study the "Precautions for Safety" on the next page before use.

Details described in this manual

CAUTION

-  For items described as "Restrictions" or "Usable State" in this manual, the instruction manual issued by the machine tool builder takes precedence over this manual.
-  Items not described in this manual must be interpreted as "not possible".
-  This manual is written on the assumption that all option functions are added. Confirm with the specifications issued by the machine tool builder before starting use.
-  Refer to the Instruction Manual issued by each machine tool builder for details on each machine tool.
-  Some screens and functions may differ depending on the NC system (or its version), and some functions may not be possible. Please confirm the specifications before use.

Refer to the following documents.

MITSUBISHI CNC 700/70 Series	Programming Manual (Machining Center System)	IB-1500072
MITSUBISHI CNC 700/70 Series	Programming Manual (Lathe System).....	IB-1500057

Precautions for Safety

Always read the specifications issued by the machine tool builder, this manual, related manuals and attached documents before installation, operation, programming, maintenance or inspection to ensure correct use.

Understand this numerical controller, safety items and cautions before using the unit.

This manual ranks the safety precautions into "DANGER", "WARNING" and "CAUTION".

DANGER

When the user may be subject to imminent fatalities or major injuries if handling is mistaken.

WARNING

When the user may be subject to fatalities or major injuries if handling is mistaken.

CAUTION

When the user may be subject to bodily injury or when physical damage may occur if handling is mistaken.

Note that even items ranked as " **CAUTION**", may lead to major results depending on the situation. In any case, important information that must always be observed is described.

DANGER

Not applicable in this manual.

WARNING

Not applicable in this manual.

CAUTION

1. Items related to product and manual

-  For items described as "Restrictions" or "Usable State" in this manual, the instruction manual issued by the machine tool builder takes precedence over this manual.
-  Items not described in this manual must be interpreted as "not possible".
-  This manual is written on the assumption that all option functions are added. Confirm with the specifications issued by the machine tool builder before starting use.
-  Refer to the Instruction Manual issued by each machine tool builder for details on each machine tool.
-  Some screens and functions may differ depending on the NC system (or its version), and some functions may not be possible. Please confirm the specifications before use.

(Continued on next page)

⚠ CAUTION

2. Items related to installation and assembly

- ⚠ Ground the signal cables to ensure stable system operation. Also ground the NC unit main frame, power distribution panel and machine to one point, so they all have the same potential.

3. Items related to preparation before use

- ⚠ Always set the stored stroke limit. Failure to set this could result in collision with the machine end.
- ⚠ Always turn the power OFF before connecting/disconnecting the I/O device cable. Failure to do so could damage the I/O device and NC unit.

4. Items related to screen operation

- ⚠ If the tool compensation amount or workpiece coordinate system offset amount is changed during automatic operation (including during single block stop), the changes will be valid from the command in next block or after several subsequent blocks.
- ⚠ All of the various data in the NC memory is erased when formatting. Be sure to use the transfer function to transfer all necessary data to another storage device before formatting.
- ⚠ Even if the tool compensation amount write command, parameter write command or variable data write command is executed with graphic check, the data will be actually written in, and the original data will be overwritten.
- ⚠ Pay close attention to the sequence operation when carrying out forced data setting (forced output) in the I/F diagnosis screen during machine operation.
- ⚠ To prevent influence from data omission and data transformation in the communication circuit, always verify the data after inputting and outputting machining programs.
- ⊘ Do not change setup parameters without prior approval from the machine tool builder.

5. Items related to programming

- ⚠ Because of key chattering, etc., during editing, "NO NOS. FOLLOWING G" commands become a "G00" operation during running.
- ⚠ " ; ", "EOB", "%", and "EOR" are symbols used for explanation. The actual codes for ISO are "CR, LF" ("LF") and "%".
The programs created on the Edit screen are stored in the NC memory in a "CR, LF" format, however, the programs created with external devices such as the FD or RS-232C may be stored in an "LF" format.
The actual codes for EIA are "EOB (End of Block)" and "EOR (End of Record)".
- ⊘ Do not change the fixed cycle program without prior approval from the machine tool builder.

(Continued on next page)

CAUTION

6. Items related to operation

-  Stay out of the moveable range of the machine during automatic operation. During rotation, keep hands, feet and face away from the spindle.
-  Carry out dry operation before actually machining, and confirm the machining program, tool offset and workpiece coordinate system offset.
-  If the operation start position is set from a block in the program and the program is started, the program before the set block is not executed. If there are coordinate system shift commands or M, S, T, and B commands before the block set as the starting position, carry out the required commands using the MDI, etc. There is a danger of interference with the machine if the operation is started from the set starting position block without carrying out these operations.
-  Program so the mirror image function is turned ON/OFF at the mirror image center. The mirror image center will deviate if the function is turned ON/OFF at a position other than the mirror image center.

7. Items related to faults and abnormalities

-  If the battery low warning is issued, save the machining programs, tool data and parameters in an input/output device, and then replace the battery. When the battery alarm is issued, the machining programs, tool data and parameters may be destroyed. Reload the data after replacing the battery.
-  If the axis overruns or emits an abnormal noise, immediately press the emergency stop button and stop the axis movement.

(Continued on next page)

CAUTION

8. Items related to maintenance

-  Incorrect connections may damage the devices, so connect the cables to the specified connectors.
-  Do not apply voltages other than those indicated according to specification on the connector. Doing so may lead to destruction or damage.
-  Do not connect or disconnect the connection cables between each unit while the power is ON.
-  Do not connect or disconnect the PCBs while the power is ON.
-  Do not connect the cable by pulling on the cable wire.
-  Do not short circuit, charge, overheat, incinerate or disassemble the battery.
-  Dispose the spent battery according to local laws.
-  Dispose the spent cooling fan according to local laws.
-  Do not replace the control unit while the power is ON.
-  Do not replace the operation panel I/O unit while the power is ON.
-  Do not replace the control section power supply PCB while the power is ON.
-  Do not replace the expansion PCB while the power is ON.
-  Do not replace the memory cassette while the power is ON.
-  Do not replace the cooling fan while the power is ON.
-  Do not replace the battery while the power is ON.
-  Be careful that metal cutting chips, etc., do not come into contact with the connector contacts of the memory cassette.
-  Do not replace the high-speed program server unit while the power is ON.
-  Before touching the unit, always touch a metal with a ground section to discharge any static electricity in the human body etc.
If static electricity is not discharged, the breakdown and the malfunction of the unit are caused.
-  When connecting to a personal computer and a unit with the RS-232/USB interface, an electric shock or a unit failure may occur.
Operate these correctly according to the manual of a unit and a personal computer. Observe the following cautions when a personal computer in an AC power supply is used.
 - (1) For a personal computer that uses a 3-pin power plug or power plug with a ground lead type, make sure to use a plug socket including a ground input electrode or ground the earth lead, respectively.
 - (2) For a personal computer that uses a 2-pin power plug without ground lead, make sure to connect the unit to the personal computer according to the following procedures.
And, it is recommended to supply the same power supply line to a personal computer and the unit.
 - (a) Pull out the power plug of the personal computer from the AC outlet.
 - (b) Confirm that the power plug of the personal computer has been pulled out from the AC outlet, and connect RS-232/USB cables.
 - (c) Insert the power plug of the personal computer into the AC outlet.

Contents

I. SCREEN OPERATIONS

1. Operating the Setting and Display Unit	1
1.1 Setting Display Unit Appearance	1
1.2 Screen Configuration	4
1.2.1 Operation Mode	5
1.2.2 MDI Status	5
1.2.3 Operation Status	6
1.2.4 Alarms/Warnings	6
1.2.5 Operation Messages	6
1.3 Screen Transition Diagram	7
1.4 Screen Selection Procedure	8
1.5 Setting Data	9
1.5.1 Setting Numerals and Alphabetic Characters	9
1.5.2 Inputting Operations	11
1.6 Operating the Screen	13
1.6.1 Changing the Displayed Part System	13
1.6.2 Changing the Menu	13
1.6.3 Types of Menus	13
1.7 Menu List	14
1.7.1 Displaying the Menu Function Outline	17
1.7.2 Directly Moving to the Menu Function	17
1.7.3 Moving the cursor	18
1.8 Guidance Function	20
1.8.1 Parameter Guidance	20
1.8.2 Alarm Guidance	25
1.9 Touch Panel Functions	28
1.9.1 Basic Operation	28
1.9.2 Menu List	30
1.9.3 Operation/Parameter Guidance	31
1.10 Touch Panel S/W Key	32
1.10.1 Operation Method	35
1.10.2 Automatic Display of S/W Keyboard [70 Series Only]	37
1.11 Screen Saver (Backlight OFF) Function	38
1.11.1 Backlight OFF	38
1.11.2 Backlight ON	39
1.12 Screen Capture [70 Series Only]	40
1.13 Multi-part System Program Management	41
1.13.1 Outline	41
1.13.2 Restrictions	42
2. Monitor Screens	43
2.1 Screen Configuration	43
2.1.1 Counter Display	50
2.1.2 Changing Between <Auto/MDI> and <Manual> [1-part System Display Only]	51
2.1.3 Operation of 2-part System Simultaneous Display	52
2.2 Operation Search	53
2.2.1 Executing an Operation Search	55
2.2.2 Changing Whether to Show or Hide the Comment Field	57
2.2.3 Changing the Sorting Method	58

2.3 Restart Search	59
2.3.1 Main Screen	61
2.3.2 Top Search Screen	63
2.3.3 File Setting Screen	64
2.3.4 MSTB History Screen	65
2.3.5 Operation Sequence for Program Restart	66
2.3.6 Executing Restart Search (Restart Type 1)	69
2.3.7 Executing Restart Search (Restart Type 2)	70
2.3.8 Changing the Device	74
2.3.9 Changing the Directory with the Main Screen	74
2.3.10 Returning to the Restart Position	75
2.3.11 Executing the MSTB Commands	77
2.4 Program Edit	78
2.5 Trace	79
2.5.1 Displaying the Machine Position Trace	82
2.5.2 Canceling the Machine Position Trace	83
2.5.3 Displaying the Machine Position of the Tool Center Point Trace	83
2.5.4 Canceling the Machine Position of the Tool Center Point Trace	83
2.5.5 Changing the Display Range	84
2.5.6 Changing the Display Mode	87
2.5.7 Changing the Display Angle	88
2.5.8 Switching the Full-screen Display Mode	89
2.5.9 Precautions for Tracing	89
2.6 Program Check (2D)	90
2.7 Program Check (3D) [700 series only]	91
2.8 Counter All-axis Display	92
2.9 Tool Compensation Amount	94
2.10 Workpiece Coordinate System Compensation	97
2.11 Counter Set	98
2.12 Origin Set, Origin Cancel	100
2.13 Manual Numerical Value Command	102
2.14 Modal Display	104
2.15 Program Tree Display	107
2.16 Integrated Time Display	108
2.16.1 Setting the Integrated Time	110
2.16.2 Setting the Time Display Selection	111
2.17 Common Variables	112
2.17.1 Setting Common Variables	115
2.17.2 Copying/Pasting Common Variables	116
2.17.3 Erasing Common Variables	117
2.18 Local Variables	118
2.18.1 Displaying the Arbitrary Local Variables	120
2.19 Buffer Correction	121
2.20 PLC Switch Function	125
2.20.1 Turning PLC Switches ON/OFF	126
2.21 Verify Stop	127
2.22 Load Meter Display	131
2.23 Spindle, Standby Display	132
2.24 Tool Center Coordinate Display	133
2.25 All Spindles' Rotation Speed Display	135

3. Setup Screens	136
3.1 Screen Configuration	136
3.2 Tool Compensation Amount.....	138
3.2.1 Tool Compensation Amount (M system).....	138
3.2.1.1 Setting the Tool Compensation Data	142
3.2.1.2 Erasing the Tool Compensation Data	144
3.2.1.3 Copying/Pasting the Tool Compensation Data	145
3.2.2 Tool Compensation Amount (L system).....	146
3.2.2.1 Wear Data	146
3.2.2.2 Tool Length Data.....	149
3.2.2.3 Tool Nose Data	151
3.2.2.4 Operation Method	154
3.3 Tool Measurement	158
3.3.1 Tool Measurement (M system)	158
3.3.2 Tool Measurement (L system)	164
3.4 Tool Registration.....	175
3.4.1 Registering a Tool in the Magazine Pot	177
3.4.2 Setting the PLC Command	178
3.4.3 Setting/Erasing the Tool No. of Spindle/Standby Tools	178
3.4.4 Erasing the Tool Registration Data	179
3.5 Tool Life Management	180
3.5.1 Displaying the Group List.....	182
3.5.2 Displaying the Life Management Data in Group Units (M system).....	186
3.5.3 Displaying the Life Management Data (L system: Tool life management I).....	194
3.5.4 Displaying the Tool Life Management Data in Group Units (L system: Tool life Management II)	197
3.6 Workpiece Coordinate System Offset.....	203
3.6.1 Setting the Coordinate System Offset.....	205
3.6.2 Erasing the Coordinate System Offset Amount	206
3.6.3 Setting the Workpiece Coordinate Origin.....	206
3.6.4 Changing the Coordinate System Display	207
3.7 Workpiece Measurement	208
3.7.1 Workpiece Measurement (M system)	208
3.7.1.1 Carrying Out Surface Measurement.....	212
3.7.1.2 Carrying Out Hole Measurement.....	213
3.7.1.3 Carrying Out Width Measurement.....	215
3.7.1.4 Carrying Out Rotation Measurement.....	217
3.7.1.5 Performing Automatic Recontact When Contacting the Workpiece	221
3.7.2 Workpiece Measurement (L system)	222
3.8 User Parameters	229
3.8.1 Selecting the Parameter Number	231
3.8.2 Setting the Parameters	231
3.8.3 Copying/Pasting Parameters	232
3.8.4 Parameter Configuration	234
3.8.5 Echoback	235
3.8.6 Ethernet Parameter Password	235
3.8.7 Machine Parameter Password Setting Method.....	236
3.9 MDI Program Editing.....	237
3.10 Counter Setting	238
3.11 Manual Numerical Value Commands.....	239
3.12 T Code List.....	240

3.13 Pallet Program Registration [700 series only]	242
3.13.1 Standard Pallet Registration	243
3.13.2 Pallet List Screen	245
3.13.3 Pallet Details Screen	247
4. Edit Screens	251
4.1 Screen Configuration	251
4.2 Program Editing	253
4.2.1 Creating a New Machining Program	256
4.2.2 Editing a Machining Program	257
4.2.3 Editing MDI Program	258
4.2.4 Registering MDI Program in NC Memory	258
4.2.5 Deleting a File	259
4.2.6 Editing Operations	259
4.2.7 Changing the Display	260
4.2.8 Displaying an Arbitrary Line	261
4.2.9 Rewriting Data	262
4.2.10 Inserting Data	263
4.2.11 Deleting Data	264
4.2.12 Searching for Character Strings	266
4.2.13 Replacing Character Strings	268
4.2.14 Copying/Pasting Data	269
4.2.15 Undoing Program Changes	270
4.2.16 Correcting/Displaying Input Mistakes [700 series only]	271
4.2.17 Adding Sequence No. (N No.) Automatically	277
4.2.18 G Code Guidance	280
4.2.19 Playback Editing	283
4.3 Program Check (2D)	288
4.3.1 Checking Continuously	292
4.3.2 Checking One Block at a Time	294
4.3.3 Canceling the Program Check	295
4.3.4 Drawing during a Program Check	295
4.3.5 Changing the Display Range	296
4.3.6 Changing the Display Mode	299
4.3.7 Changing the Display Angle	300
4.3.8 Switching to Full-screen Display Mode	301
4.3.9 Availability of Check Mode with Other Functions	302
4.3.10 Handling of Variable Commands, Parameter Input by Program, and Compensation Data Input by Program	303
4.3.11 Notes for Program Check (2D)	304
4.4 Program Check (3D) [700 series only]	306
4.4.1 Checking Continuously	309
4.4.2 Checking One Block at a Time	310
4.4.3 Canceling the Program Check	310
4.4.4 Enlarging and Reducing the Workpiece Shape	311
4.4.5 Moving the Workpiece Shape	311
4.4.6 Rotating the Workpiece Shape	311
4.4.7 Performing an Interference Check	312
4.4.8 Switching to Full-screen Display Mode	312
4.4.9 Setting the Workpiece Shape	313
4.4.10 Setting the Tool Shape	318
4.4.11 Availability of Check Mode with Other Functions	323
4.4.12 Handling of Variable Commands, Parameter Input by Program, and Compensation Data Input by Program	323
4.4.13 Notes for Program Check (3D)	323

4.5 Program Input/Output.....	326
4.5.1 Changing the Valid Area	330
4.5.2 Selecting a Device, Directory, and File	331
4.5.3 Transferring a File	335
4.5.4 Comparing Files (Compare).....	337
4.5.5 Erasing a File	338
4.5.6 Changing a File Name (Rename)	339
4.5.7 Creating a Directory	340
4.5.8 Merging a File	341
4.5.9 Formatting an External Device.....	342
4.5.10 List of File Names	342
4.5.11 Edit Lock B and C	343
4.5.12 Program Display Lock C	344
4.5.13 Data Protect Keys	345
4.5.14 Sharing Machining Data.....	346
4.5.15 The Batch Input/Output the Machining Program of NC Memory.....	349
5. Diagnosis Screens.....	353
5.1 System Configuration Screen	353
5.2 Option Display Screen	356
5.3 I/F Diagnosis Screen.....	357
5.3.1 Displaying the PLC Device Data	360
5.3.2 Carrying Out Modal Output	361
5.3.3 Carrying Out One-shot Output	362
5.4 Drive Monitor Screen	363
5.4.1 Servo Axis Unit Display Items	365
5.4.2 Spindle Unit Display Items	368
5.4.3 Display Items for the Power Supply Unit.....	377
5.4.4 Display Items for the Auxiliary Axis Unit [700 series only]	379
5.4.5 Display Items for the Synchronous Error	381
5.4.6 Clearing the Alarm History	382
5.5 NC Memory Diagnosis Screen (NC Memory Diagn Screen)	383
5.5.1 Writing/Reading the Data Using the NC Data Designation	385
5.6 Alarm Screen	386
5.6.1 Alarm History.....	388
5.7 Self Diagnosis Screen.....	390
5.8 Data Sampling Screen	394
5.8.1 Executing NC Data Sampling.....	399
5.8.2 Outputting Sampling Data	400
5.9 Anshin-net Screen.....	402
5.9.1 Message Display Screen	402
5.9.2 Anshin-net Parameter 1, 2 Screen.....	404
5.9.3 Operation Method	405
5.9.4 Sharing Machining Data.....	406
5.10 Machine Tool Builder Network System (MTB net) Screen.....	407
5.10.1 Message Display Screen	407
5.10.2 MTB Net Parameter 1, 2 Screen	409
5.10.3 Operation Method	410

6. Maintenance Screens	411
6.1 Parameter Screens	413
6.1.1 Changing the Parameter Display	418
6.1.2 Setting the Parameters	419
6.1.3 Copying/Pasting Parameters	421
6.1.4 User Parameters	423
6.1.5 Echo Back	424
6.2 Input/Output Screen	425
6.2.1 Changing the Valid Area	428
6.2.2 Selecting a Device, Directory and File	429
6.2.3 Transferring a File	436
6.2.4 Comparing Files (Compare)	438
6.2.5 Erasing a File	439
6.2.6 Changing a File Name (Rename)	439
6.2.7 Creating a Directory	440
6.2.8 Merging a File	441
6.2.9 Formatting an External Device	442
6.2.10 List of File Names	443
6.2.11 Edit Lock B and C	452
6.2.12 Data Protect Keys	453
6.2.13 Sharing Machining Data	455
6.2.14 The Batch Input/Output the Machining Program of NC Memory	458
6.3 All Backup Screen	462
6.3.1 Performing a Backup Operation	464
6.3.2 Performing a Restore Operation	464
6.4 System Setup Screen	465
6.4.1 Preparation for Spindle Parameter Setting	467
6.4.2 Writing Initial Parameters	468
6.4.3 Writing Sample PLC Ladders	469
6.5 Adjust S-analog Screen	470
6.5.1 Adjustment Preparations	472
6.5.2 Performing Automatic Adjustments	472
6.5.3 Performing Manual Adjustments	473
6.6 Absolute Position Setting Screen	475
6.6.1 Selecting the Axis	477
6.6.2 Carrying Out Dogless-type Zero Point Initialization	478
6.6.3 Carrying Out Dog-type Zero Point Initialization	486
6.6.4 Precautions	486
6.7 Auxiliary Axis Test Screen	489
6.7.1 Preparation	492
6.7.2 Selecting a Device, Directory and File	494
6.7.3 Test Operation	500
6.7.4 Precaution	500
6.8 Diagnosis Data Collection Setting Screen	501
6.8.1 Performing a Data Collection Operation	503
6.8.2 Stopping a Data Collection Operation	503
6.8.3 Clearing a Collected Data	503

II. MACHINE OPERATIONS

1. Operation State	2
1.1 Operation State Correlation Diagram	2
1.2 Power OFF	2
1.3 Operations Not Ready	3
1.4 Operations Ready	3
1.4.1 Reset	3
1.4.2 Automatic Operation in Progress	3
1.4.3 Automatic Operation Pause	3
1.4.4 Automatic Operation Stop	3
2. Indicator Lamps	4
2.1 Control Unit Ready	4
2.2 In Automatic Operation	4
2.3 Automatic Operation Start Busy	4
2.4 Automatic Operation Pause Busy	4
2.5 Return to Reference Position	4
2.6 Alarm	4
2.7 M00	4
2.8 M02/M30	4
3. Reset Switch and Emergency Stop Button	5
3.1 Reset Switch	5
3.2 Emergency Stop Button	5
4. Operation Mode	6
4.1 Mode Selection Switch	6
4.2 Jog Feed Mode	6
4.3 Rapid Traverse Feed Mode	7
4.4 Manual Reference Position Return Mode	8
4.5 Incremental Feed Mode	10
4.6 Handle Feed Mode	10
4.7 Memory Mode	11
4.8 MDI Operation Mode	12
5. Operation Panel Switches in Operation Mode	13
5.1 Rapid Traverse Override	13
5.2 Cutting Feed Override	13
5.3 Spindle Override	13
5.4 Manual Feedrate	14
5.5 Handle/Incremental Feed Magnification	14
5.6 Handle Feed Axis Selection	14
5.7 Manual Pulse Generator	15
5.8 Cycle Start and Feed Hold	15
5.9 Feed Axis Selection	15
6. Operation Panel Switch Functions	16
6.1 Chamfering	16
6.2 Miscellaneous Function Lock	16
6.3 Single Block	16
6.4 Z Axis Cancel	16
6.5 Dry Run	16
6.6 Manual Override	16
6.7 Override Cancel	17
6.8 Optional Stop	17
6.9 Optional Block Skip	17

6.10 Manual Absolute	18
6.11 Mirror Image.....	19
6.11.1 Outline.....	19
6.11.2 Detailed Description	20
6.11.3 Combination with Other Functions	22
6.11.4 Precautions	25
6.12 Error Detect.....	27
6.13 Follow-up Function.....	27
6.14 Axis Removal	27
6.15 F 1-digit Feed.....	28
6.16 Manual/Automatic Synchronous Feed	29
6.16.1 Outline.....	29
6.16.2 Conditions for Validating Manual/Automatic Synchronous Feed	29
6.16.3 Operation during manual/automatic simultaneous operation.....	30
6.16.4 Operation When Automatic Operation Commands and Manual/Automatic Simultaneous Valid Selection are Issued for Same Axis	31
6.16.5 Other Precautions	33
6.17 Handle Interruption.....	34
6.17.1 Outline.....	34
6.17.2 Interruptible Conditions	34
6.17.3 Interruption Effective Axis	34
6.17.4 Axis Movement Speed Resulting From Interruption.....	35
6.17.5 Path Resulting After Handle Interruption.....	35
6.17.6 Handle Interruption in Tool Radius Compensation	37
6.17.7 Interrupt Amount Reset.....	39
6.17.8 Operation Procedure.....	39
6.18 All Axis Machine Lock	40
6.19 Each Axis Machine Lock	40
6.20 Deceleration Check.....	41
6.20.1 Functions.....	41
6.20.2 Deceleration Check Method.....	42
6.20.3 Deceleration Check When Opposite Direction Movement Is Reversed.....	44
6.20.4 Parameters.....	45
6.20.5 Precautions	46
6.21 Tool Return and Escape	48
6.21.1 Outline.....	48
6.21.2 Retracting Method.....	48
6.21.3 Example of Operation	51
6.21.4 Precautions	52
6.22 External Deceleration.....	53
6.22.1 Outline.....	53
6.22.2 Explanation of Function.....	53
6.22.3 Combination with Other Functions	53
6.22.4 Precautions	53
6.23 Reference Position Retract	54
6.23.1 Outline.....	54
6.23.2 Detailed Description	54
6.24 Spindle Orientation.....	55
6.24.1 Outline.....	55
6.24.2 Orientation Operation.....	56

7. Other Functions	59
7.1 Stored Stroke Limit.....	59
7.1.1 Stored Stroke Limit I.....	61
7.1.2 Stored Stroke Limit II.....	62
7.1.3 Stored Stroke Limit IB	64
7.1.4 Stored Stroke Limit IC	64
7.1.5 Movable Range during Inclined Axis Control	65
7.1.6 Stored Stroke Limit for Rotation Axis	66
7.1.7 Precautions	67
7.2 Chuck Barrier/Tailstock Barrier (L System).....	68
7.2.1 Detailed Description	68
7.2.2 Setting the Chuck Barrier/Tailstock Barrier	69
7.2.3 Restrictions	72
7.3 Computer Link B	73
7.3.1 Communication Procedure.....	74
7.4 Manual Synchronous Tapping	76
7.4.1 Outline.....	76
7.4.2 Command Format	76
7.4.3 Operation Procedures	76
7.4.4 Precautions	76

III. MAINTENANCE

1. Daily Maintenance and Periodic Inspection and Maintenance	1
1.1 Maintenance Items.....	1
1.1.1 Escutcheon	1
1.1.2 LCD Panel.....	2
1.1.3 Compact Flash/IC card	2
1.2 Replacement Methods [700 series].....	3
1.2.1 Durable Parts	3
1.2.2 Compact Flash	7
1.2.3 IC Card.....	8
1.2.4 How to Replace the Protective Sheet on the Touch Panel	9
1.3 Replacement Methods [70 series].....	10
1.3.1 Durable Parts	10
1.3.2 Compact Flash	12
1.3.3 How to Replace the Protective Sheet on the Touch Panel	12

IV. APPENDIXES

Appendix 1. List of Function Codes.....	1
Appendix 2. Table of Command Value Ranges.....	2
Appendix 3. Circular Cutting Radius Error.....	8

Appendix 4. Registering/Editing the Fixed Cycle Program.....	9
4.1 Fixed Cycle Operation Parameters	9
4.2 Transmitting/Erasing the Fixed Cycle Program.....	9
4.3 Standard Fixed Cycle Subprogram (For L system).....	10
4.4 Standard Fixed Cycle Subprogram (For M system).....	20
Appendix 5. RS-232C I/O Device Parameter Setting Examples	26
Appendix 6. Alarms	27
6.1 List of Alarms	27
6.1.1 Operation Alarms	27
6.1.2 Stop Codes	36
6.1.3 Servo/Spindle Alarms.....	41
6.1.4 MCP Alarm.....	51
6.1.5 System Alarms	61
6.1.6 Absolute Position Detection System Alarms	67
6.1.7 Distance-coded Reference Scale Errors	70
6.1.8 Messages during Emergency Stop	71
6.1.9 Auxiliary Axis Alarms.....	73
6.1.10 Computer Link Errors	80
6.1.11 User PLC Alarms	81
6.1.12 Network Service Errors	83
6.2 Operation Messages.....	84
6.2.1 Search-related Operation Messages	84
6.2.2 Graphic Display-related Operation Messages	85
6.2.3 Variable (Common variables, local variables) - related Operation Messages	86
6.2.4 PLC Switch-related Operation Messages	86
6.2.5 Compensation-related (Tool compensation, coordinate system offset) Operation Messages	86
6.2.6 Data Input/Output-related Operation Messages	87
6.2.7 Parameter-related Operation Messages	90
6.2.8 Measurement-related (Workpiece, rotation) Operation Messages	91
6.2.9 Tool (Tool registration, tool life) -related Operation Messages	93
6.2.10 Editing-related Operation Messages.....	94
6.2.11 Diagnosis-related Operation Messages	96
6.2.12 Maintenance-related Operation Messages	97
6.2.13 Data Sampling-related Operation Messages	99
6.2.14 Absolute Position Detection-related Operation Messages.....	100
6.2.15 System Setup-related Operation Messages	100
6.2.16 Automatic Backup-related Operation Messages.....	101
6.2.17 Alarm History-related Operation Messages	101
6.2.18 Anshin-net-related Operation Messages.....	102
6.2.19 Messages Related to Machine Tool Builder Network System	107
6.2.20 Other Operation Messages	109
6.3 Program Error	110
Appendix 7. G Code Guidance Display List	130
Appendix 8. IP Address Resetting Procedure at Disabled Network Communication [700 Series Only].....	137
8.1 Connectable Control Unit IP Address List Screen	137
8.2 Resetting Procedure	138
8.3 Message.....	139

Appendix 9. User Parameter List.....	140
9.1 Process Parameters.....	140
9.2 Fixed Cycle	148
9.3 Control Parameters 1	151
9.4 Control Parameters 2	157
9.5 I/O Parameters.....	159
9.6 Ethernet Parameters	178
9.7 Computer Link Parameters	186
9.8 Subprogram Storage Destination Parameters	189
9.9 Axis Parameters.....	193
9.10 Barrier Data (For L System Only).....	196
9.11 High Accuracy Parameters	199
9.12 High-accuracy Axis Parameters.....	204
9.13 Operation Parameters.....	205

I. SCREEN OPERATIONS

1. Operating the Setting and Display Unit

This explains the functions common to the screens.

1.1 Setting Display Unit Appearance

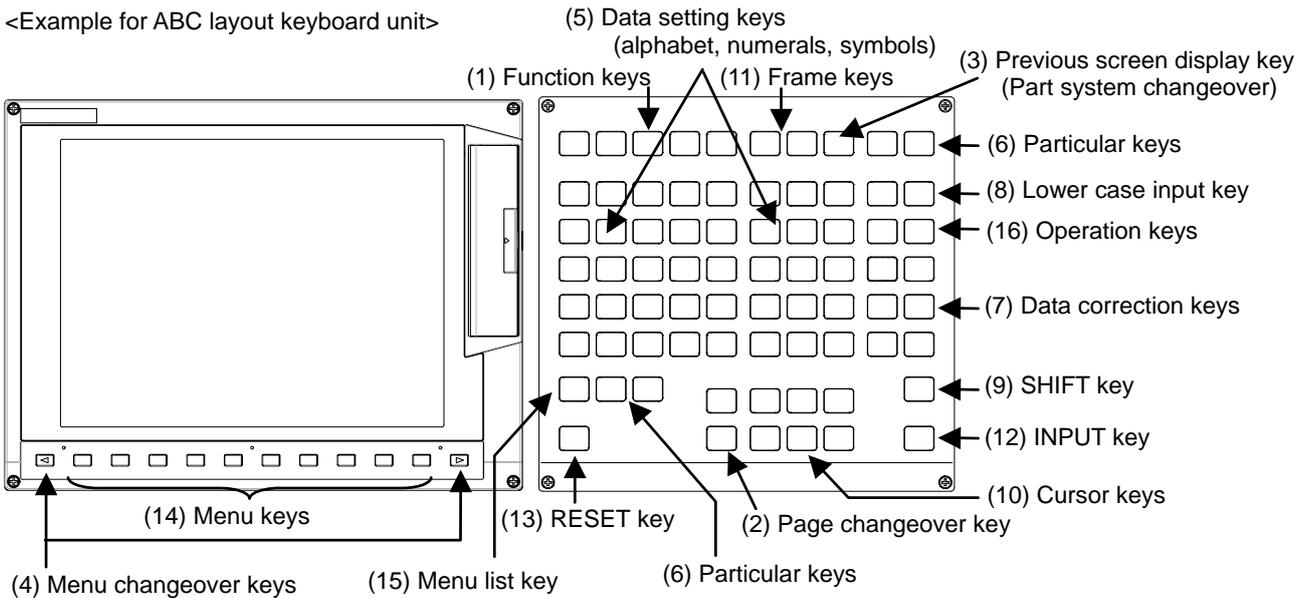
An LCD display is used for the screen displays.

Operations such as screen transition and data setting are carried out with the NC keyboard.

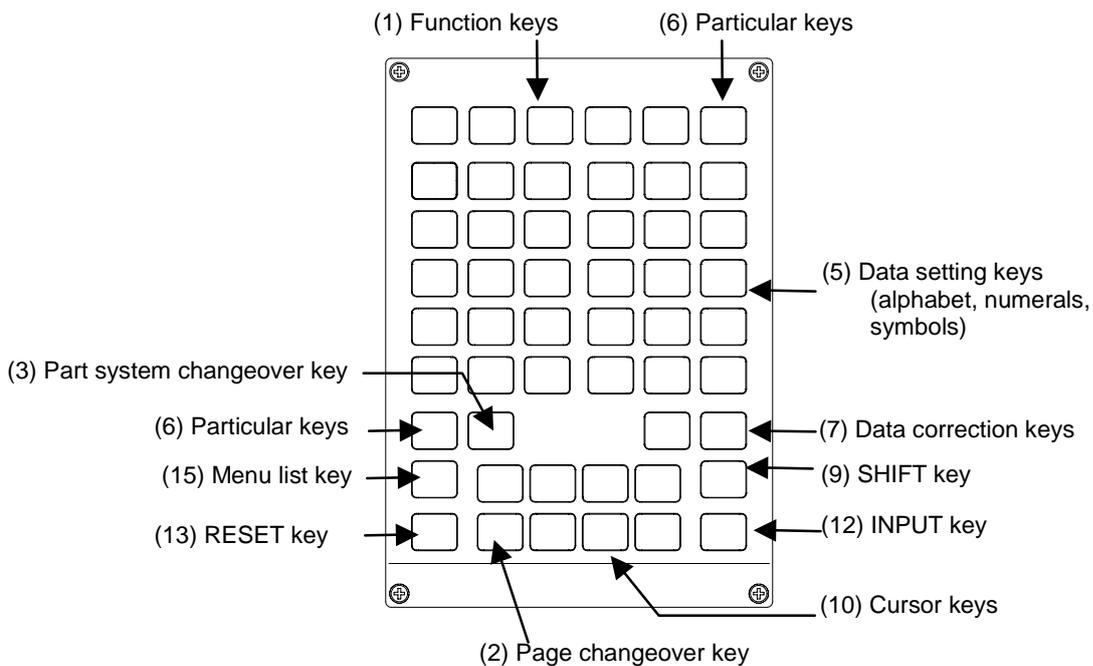
The setting display unit is configured of the LCD display, various keys and menu keys as shown below.

The drawing below shows a horizontal layout of the LCD display and NC keyboard, but these can also be arranged vertically.

<Example for ABC layout keyboard unit>



There is the following type of keyboard, too.

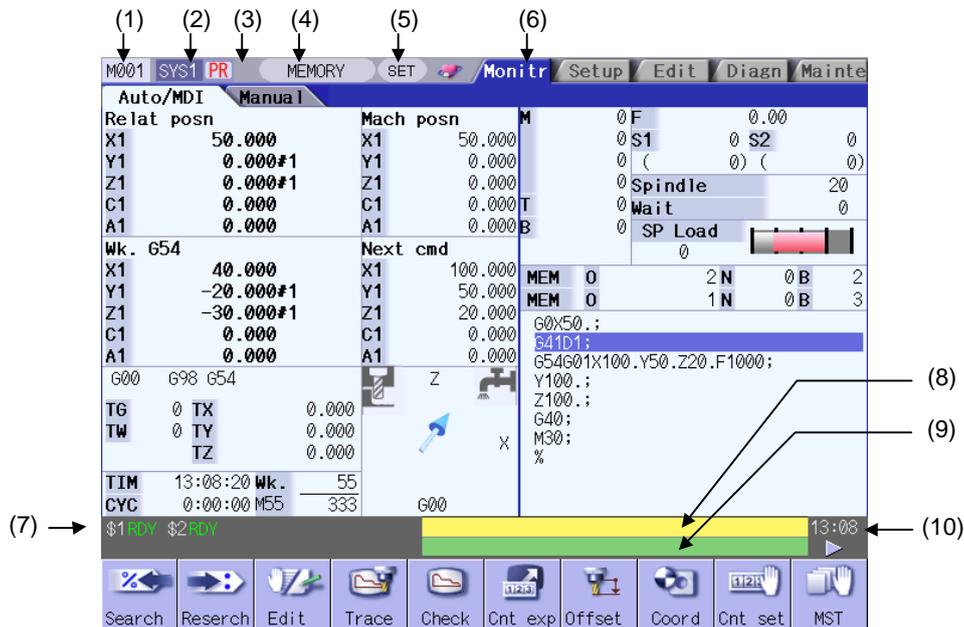


The following keys are provided on the keyboard.

	Key type	Key	Operation																																																
(1)	Function key (Function selection key)	 (MONITOR)	This displays the screen related to "operations". (Refer to "2. Monitor Screens".)																																																
		 (SETUP)	This displays the screen related to "setup". (Refer to "3. Setup Screens".)																																																
		 (EDIT)	This displays the screen related to "editing". (Refer to "4. Edit Screens".)																																																
		 (DIAGN)	This displays the screen related to "diagnosis". (Refer to "5. Diagnosis Screens".)																																																
		 (MAINT)	This displays the screen related to "maintenance". (Refer to "6. Maintenance Screens".)																																																
(2)	Page changeover key	 Previous page key	When the displayed contents cover several pages, this displays the contents of the previous page. The "▲" mark at the top of the screen indicates that there is a previous page.																																																
		 Next page key	When the displayed contents cover several pages, this displays the contents of the next page. The "▼" mark at the top of the screen indicates that there is a next page.																																																
(3)	Previous screen display key (Part system changeover)	 (BACK) Previous screen display key	This redisplay the previously displayed screen.																																																
		 (\$→\$) Part system changeover key	When using a multi-part system NC, this displays the data of the next part system. The screen does not change if it is a part system common screen or when only one part system is used.																																																
(4)	Menu changeover key	 (left side)	This changes the operation menu for the displayed screen to the current screen group screen selection menu. This is also used to cancel the menu operations of the displayed screen.																																																
		 (right side)	When all of the menus cannot be displayed at once, this displays the menus not currently displayed. The "◀" and "▶" marks at the bottom of the screen indicate that there are menus not displayed.																																																
(5)	Data setting keys	<table border="1" style="border-collapse: collapse; text-align: center;"> <tr><td>A</td><td>B</td><td>C</td><td>D</td><td>E</td><td>F</td></tr> <tr><td>G</td><td>H</td><td>I</td><td>J</td><td>K</td><td>L</td></tr> <tr><td>M</td><td>N</td><td>O</td><td>P</td><td>Q</td><td>R</td></tr> <tr><td>S</td><td>T</td><td>U</td><td>V</td><td>W</td><td>X</td></tr> <tr><td>Y</td><td>Z</td><td></td><td></td><td></td><td></td></tr> <tr><td>0</td><td>1</td><td>2</td><td>3</td><td>4</td><td>5</td></tr> <tr><td>6</td><td>7</td><td>8</td><td>9</td><td>+</td><td>-</td></tr> <tr><td>=</td><td>/</td><td>.</td><td>;</td><td colspan="2">etc.</td></tr> </table>	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	R	S	T	U	V	W	X	Y	Z					0	1	2	3	4	5	6	7	8	9	+	-	=	/	.	;	etc.		These keys are pressed to set alphabetic characters, numerals and operation symbols, etc.
A	B	C	D	E	F																																														
G	H	I	J	K	L																																														
M	N	O	P	Q	R																																														
S	T	U	V	W	X																																														
Y	Z																																																		
0	1	2	3	4	5																																														
6	7	8	9	+	-																																														
=	/	.	;	etc.																																															

	Key type	Key	Operation
(6)	Particular keys	 Help key	This displays the operation guidance, parameter guidance and alarm guidance corresponding to the current operation.
		 / 	These key definitions differ according to the machine tool builder.
			
			
			
(7)	Data correction keys	 (INSERT) Data insert key	This inputs the data insertion mode. When a data setting key is pressed, a character is inserted in front of the current cursor position. The overwrite mode is entered when the DELETE , C·B CAN , INPUT , cursor or TAB, etc., keys are pressed, or when the screen is changed.
		 (DELETE) Data delete key	This deletes the character just before the cursor position in the data setting area.
		 /   (C·B CAN) Cancel key	This cancels the setting in the data setting area.
(8)	Lower case input key	 (LOWER CASE)	This changes the input between upper case and lower case alphabetic characters.
(9)	SHIFT key	 (SHIFT)	This validates the setting on the lower line of data key.
(10)	Cursor keys	 	This moves the cursor up or down one when setting data in the screen display items.
		 	This moves the cursor one item to the left or right when selecting data in the screen display items.  at cursor left end: Moves to the right end of previous line.  at cursor right end: Moves to left end of next line.
		 	This moves the data input cursor one character to the left or right in the data setting area.
(11)	Frame keys	 	This switches the tag.
(12)	INPUT key	 (INPUT)	This fixes the data in the data setting area, and writes it to the internal data. The cursor moves to the next position.
(13)	RESET key	 (RESET)	This resets the NC.
(14)	Menu keys		This changes the screen and displays the data.
(15)	Menu list keys	 (MenuList)	This is function that displays each screen's menu configuration as a list. (Refer to "1.7 Menu list")
(16)	Operation keys	 (ALTER)	This is alternate key (Alt key).
		 (CTRL)	This is control key.
		 (SP)	This is space key.

1.2 Screen Configuration



Display items

Display item	Details
(1) NC name	The currently displayed NC name (name set in parameter "#1135 unt_nm") is displayed.
(2) Part system name	When using the multi-part system, the currently displayed part system name (name set in parameter "#1169 system name") is displayed. The part system name is not displayed for the 1-part system.
(3) Power ON request	This appears if a parameter requiring the power to be turned ON again has been changed. This flickers at an approx. one-second interval.
(4) Operation status	The displayed part system's operation mode is displayed.
(5) MDI status	The MDI status is displayed when the MDI operation mode is selected. This does not appear when other operation modes are selected.
(6) Screen name	The tag for the currently selected screen is selected and displayed.
(7) Operation status	The NC operation status is displayed.
(8) Alarm	The currently occurring alarm or warning with the highest priority is displayed.
(9) Operation message	The operation message is displayed.
(10) Time	The current time is displayed. (Hour: minute)
(11) Anshin-net status	This icon appears if Anshin-net diagnosis data has been sent/received.

1.2.1 Operation Mode

The operation status indicates the operation mode for displayed part system.
The selectable operation mode is following below.

Symbol	Details	Explanation
MEMORY	Memory operation	Automatic operation is based on programs stored in the memory.
TAPE	Tape operation	Automatic operation is based on tape command (RS232C input) programs stored in NC tape.
MDI	MDI operation	Automatic operation is performed with the program set in the MDI screen.
JOG	Jog	The jog feed mode enables the axis to be moved by hand consecutively at the feed rate set by using the MANUAL FEED RATE switch.
HANDLE	Manual handle	The handle feed mode enables the axis to be moved by turning the manual handle. The travel distance per graduation of the handle depends on how the HANDLE/INCREMENTAL MAGNIFICATION switch is set.
STEP	Step	The step feed mode enables the axis to be moved by hand at feed rate when the FEED AXIS SELECT switch is ON. The travel distance per graduation of the handle depends on how the HANDLE/INCREMENTAL MAGNIFICATION switch is set.
MANUAL	Manual arbitrary feed	This mode enables to move manually with arbitrary amount or arbitrary positioning position.
ZERO-RTN	Reference position return	This mode enables a controlled axis to be returned manually to the defined position unique to the machine (reference position).
INIT-SET	Automatic dog-less reference position return	This mode enables this mode to manually push against the machine end stopper and carry out zero point return.
RAPID	Rapid traverse	The rapid traverse feed mode enables the machine to be moved consecutively at rapid traverse feedrate manually.
NO MODE	No operation mode	The operation mode is not selected.

(Note 1) Note the following points when using two or more part systems.

The operation mode displays the operation mode selected for the 1st part system. Thus, when using two-or-more-part-system machine for which the operation modes are separately selected for each part system, the operation mode displayed for a part system other than the 1st part system, and the operation mode on the machine may not match.

1.2.2 MDI Status

The MDI status is displayed when the operation mode is the MDI mode.
This does not appear when other operation modes are selected.
The displayed MDI status is following below.

Symbol	Details	Character color	Background color
NON	No MDI setting	Black	Gray
SET	MDI setting completed	Black	Gray
RUN	MDI running	Black	Gray

1.2.3 Operation Status

1 HLD 2 HLD 3 HLD 4 BST

The operation status displays the currently selected NC operation status for each part system. (The status for up to four part systems is displayed.)

Symbol	Details	Character color	Background color
EMG	In emergency stop	Red	Dark gray
RST	Resetting NC	White	Dark gray
LSK	Paper tape reader is in label skip state	White	Dark gray
BST	In block stop	White	Dark gray
HLD	Operation halted	White	Dark gray
SYN	Synchronizing	White	Dark gray
AUT	In automatic operation	White	Dark gray
RDY	Operation completed state	Green	Dark gray

1.2.4 Alarms/Warnings

When an alarm or warning occurs, the alarm No. and alarm message character string are displayed.

(Example) Warning display

S51 Parameter error

	Character color	Background color
Alarm	White	Red
Warning	Black	Yellow

The display format of PLC alarms and operator messages can be selected by the parameter "#11021 PLC mesg disp type".

0: Display up to the first 40 characters.

1: If text is longer than 40 characters, divide it into two and display separately. (Classification No. is displayed together)

1.2.5 Operation Messages

	Character color	Background color
Operation messages	Black	Green

The operation message can be reset by pressing any key.

The following operation messages are released even if automatic start or reset is executed.

- "Search completed"
- "Restart search is completed"
- "Top search completed"
- "MDI Set ended"

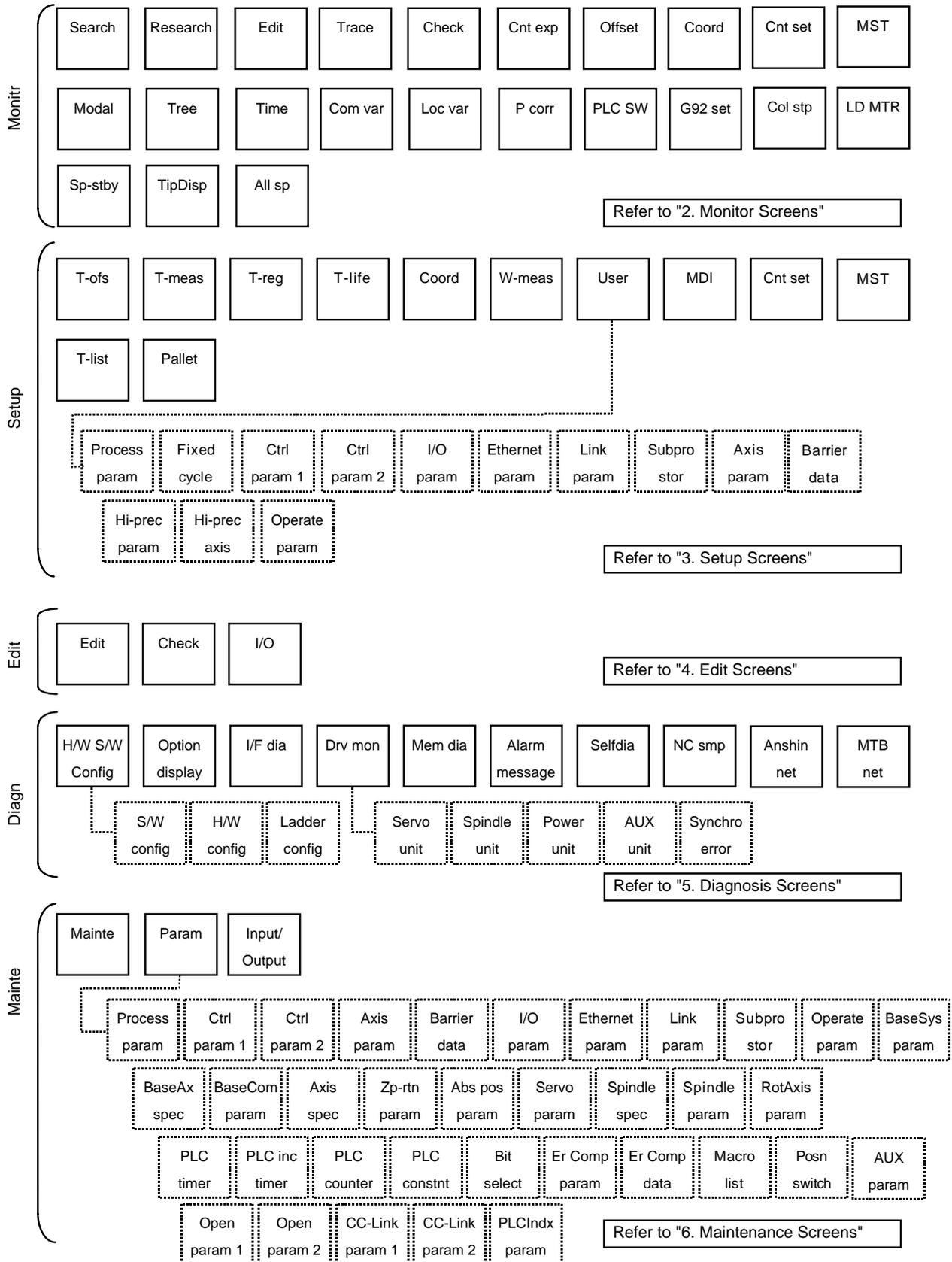
(Example)

Searching

1.3 Screen Transition Diagram

The screen is configured of operation groups.

Refer to "1.4 Screen Selection Procedures" for how to display the screens.



1.4 Screen Selection Procedure

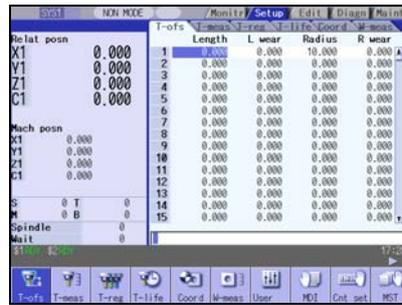
The screen is selected by pressing a function key such as **MONITOR** or **SETUP**, or by pressing a menu key displayed in the screen selection menu.

Operation method (To display "T-ofs" screen from the "Setup screen")

(1) Press the function key **SETUP**.



The previously displayed setup screen will appear.

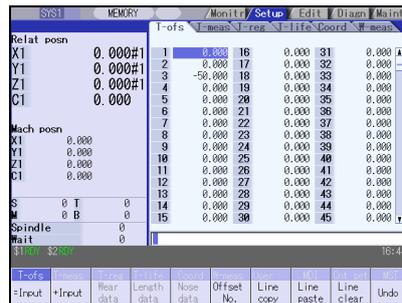


The menu display will differ according to the L system/M system and parameter setting.

(2) Press the menu **T-ofs** key.



Tool compensation amount screen appears.



1.5 Setting Data

1.5.1 Setting Numerals and Alphabetic Characters

Operation method

The data is basically set with the following methods:

- (1) Menu selection
- (2) No. selection
- (3) Cursor movement
- (4) Data key input
- (5) **INPUT** key input

(Note 1) The contents in the data setting area are only displayed until the **INPUT** key is pressed.

These contents are invalidated if the screen is changed. The data is written into the memory when the **INPUT** key is pressed.

(Note 2) Special settings may be required depending on the data type. Refer to each item.

(Note 3) The cursor may move to the right of the display item depending on the data type.

(Note 4) If an illegal key is set, an error occurs when **INPUT** is pressed. Reset the correct data.

Operations in the data setting area

The key is input at the position where the cursor is displayed. If a cursor is not displayed, the key input is invalid.

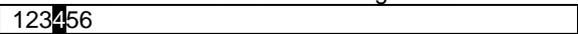
When a key is input, the data appears at the cursor position, and the cursor moves one character space to the right.

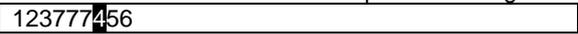
- **→/←** keys: Moves the cursor one character to the left or right.

(1) The cursor is at the position shown on the right. 

(2) Press the **→** key.  The cursor moves one character space to the right.

- **INSERT** key: Enters the insert mode.

(1) Move the cursor to the position where the data is to be inserted.  The cursor moves in the data setting area.

(2) Press the **INSERT** key, and then the data keys.  The cursor moves one character space to the right.

(Note) The overwrite mode is entered when the **DELETE**, **C-B CAN** keys are pressed, or when the screen is changed.

- **DELETE** key: Deletes the character in front of the cursor.

(1) Move the cursor to the position where the data is to be deleted.  The cursor moves in the data setting area.

123777456

(2) Press the **DELETE** key.  The character in front of the cursor is deleted, and the cursor moves.

12377456

- **C·B** key: Deletes all characters in the data setting area.

(1) Press the **C·B** key.  The cursor moves in the data setting area.

█

Cursor operations on the screen

If a cursor is displayed on the screen, data is set in the data setting area and the **INPUT** key is pressed, the data appears at the cursor position on the screen. The cursor moves to the next position. The following keys can be used to move the cursor with the cursor keys.

-  : Moves the cursor to the previous line.
-  : Moves the cursor to the next line.
-  : Moves the cursor one item to the left.
-  : Moves the cursor one item to the right.

1.5.2 Inputting Operations

In addition to the method of directly inputting numeric data for specific data settings, a method to input the operation results using four rules operators and function symbols can be used.

Input method

Numeric values, function symbols, operators and parentheses () are combined and set in the data setting area.

The operation results appear when the **INPUT** key is pressed. Data for the currently selected setting item will be set when **INPUT** key is pressed again. The contents in the data setting area are erased.

Examples of operator settings, and results			Function symbols, setting examples and results			
Operation	Setting example	Operation results	Function	Function symbol	Setting example	Operation results
Addition	=100+50	150	Absolute value	ABS	=ABS (50-60)	10
Subtraction	=100-50	50	Square root	SQRT	=SQRT (3)	1.7320508
Multiplication	=12.3*4	49.2	Sine	SIN	=SIN (30)	0.5
Division	=100/3	33.3333333	Cosine	COS	=COS (15)	0.9659258
Function	=1.2*(2.5+SQRT(4))	5.4	Tangent	TAN	=TAN (45)	1
			Arc tangent	ATAN	=ATAN (1.3)	52.431408

Operation examples

- (1) Set as shown below, and press the **INPUT** key.
 =12*20 **INPUT** → The operation results appear in the data setting area.
 240
- (2) Press the **INPUT** key again. → Data for the selected setting item is set. The result is displayed on the screen. The cursor moves to the next position.

Notes for using operators and functions

- Division : Zero division causes an error.
 Square root : If the value in the parentheses is negative, an error occurs.
 Triangle function : The unit of angle θ is degree ($^{\circ}$).
 Arc tangent : $-90 < \text{operation results} < 90$.

Restrictions

- (1) Always use "=" for the first character.
- (2) Do not use the following characters as the second character or last character.
Invalid as second character: *, /,)
Invalid as last character: *, /, (, +, -
- (3) Make sure that the left parentheses and right parentheses are balanced.
- (4) The 360° limit does not apply on the angle. SIN (500) is interpreted as SIN (140).
- (5) While the Monitor screen or the Setup screen displaying, if the axis name address, M, S, T, 2nd miscellaneous function code (B, etc.) is pressed when "=" is set in data setting area as the first character, the operation input is given to priority. So, the window does not open, and the cursor does not move.
- (6) The exponential setting, like "1.23E-4", cannot be used. The operation result is not displayed with exponential.
- (7) It is not possible to set characters exceeding the number of characters which can be input to the data setting area.
- (8) It is not possible to omit "0" before the decimal point, like ".5", when operation inputting. An error will occur.
- (9) The accuracy is guaranteed for the calculation with 15 digits or less. The calculation with over 15 digits does not guarantee its accuracy.
- (10) The operator or the function which does not exist in the above-mentioned, such as "ASIN", cannot be used. The operation message "Setting error" appears, and the display of the data setting area does not change.
- (11) Regardless of the input setting unit and metric system/inch system, the digit numbers below the decimal point of the operation result is seven digits or less.

1.6 Operating the Screen

1.6.1 Changing the Displayed Part System

When the  key is pressed, the displayed part system changes.

The displayed part system No. is counted up by one each time the  key is pressed. If the displayed part system No. exceeds the value number of part systems, the displayed part system No. returns to 1.

1.6.2 Changing the Menu

The menu can be used to select screens and to select functions or setting items. Up to ten menus can be displayed at once.

To select the menu, use the menu key below the menu display.

To change the menu, use the menu changeover key.

-  key : The operation menu is canceled.
The screen selection menu for the currently displayed screen group appears.
The display for the currently displayed menu is highlighted.
-  key : When there are more than 11 menus, this key displays the remaining menus.
This key can be used when "◀" or "▶" is displayed at the upper right of the menu.

1.6.3 Types of Menus

The menus are categorized into the following types according to the operation that takes place after the menu key is pressed.

A: The menu is highlighted, and the system waits for the user input. After the input, the operation follows the input details.

B: The menu is highlighted, and operation starts.

C: Operation starts without highlighting the menu.

In section "2. Monitor screen" and following, the types are described in the following method.

(Example) Explanation of menus used for editing the program on the Edit screen (excerpt)

Menu	Details	Type	Reference
	When the character string is designated and the  key is pressed, that character string is searched for.	A	4.2.11 Searching for character strings
	If the character string to be searched for and the character string to be replaced are separated with a "/" and designated, when the  key is pressed, the replace operation takes place.	A	4.2.12 Replacing character strings

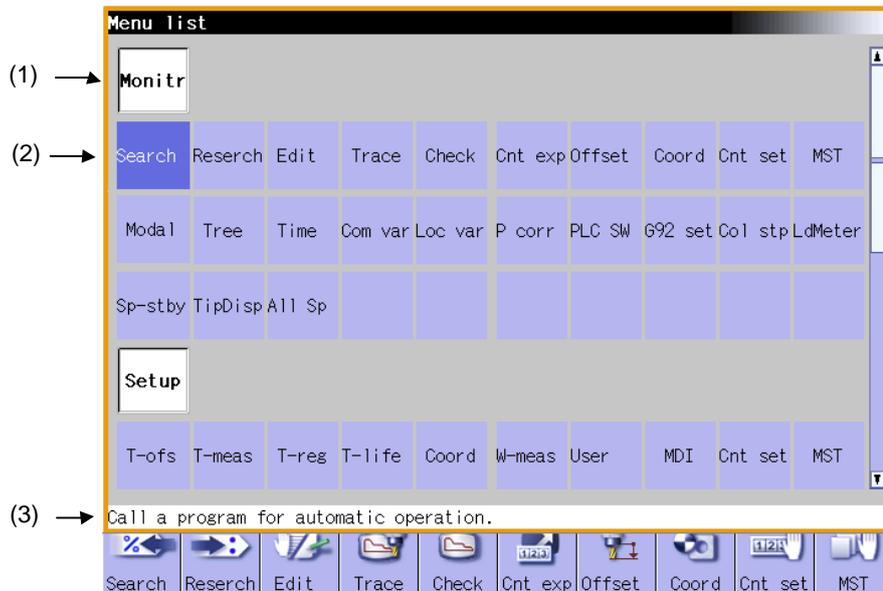
Operation when menu is pressed
Menu type
Reference section

1.7 Menu List

The menu list is a function that displays each screen's menu configuration as a list. The Menu list window opens when the **MenuList** key is pressed on each screen.

If a pop-up window other than the menu list is displayed, the Menu list window appears above the displayed pop-up window. The menu does not change in this case.

If the **MenuList** key is pressed again or if the **Cancel** key is pressed while the Menu list window is open, the Menu list window closes, and the state before the **MenuList** key was pressed is recovered.



Display items

Display item	Details
(1) Screen name	The screen name is displayed. (Example) Monitr/Setup
(2) Menu name	A list of the menu names (functions) included on each screen is displayed. (Example) Search/Reserch
(3) Function outline display area	An outline of the currently selected menu name (function) is displayed.

List of menu names (functions)

Screen	Menu name	Outline
Monitr	Search	Call a program for automatic operation.
	Reserch	Restart machining from a selected block.
	Edit	Edit the machining program searched for operation.
	Trace	Trace the T path based on the machining program being executed.
	Check	Trace the T path based on the machining program w/o running machine.
	Cnt exp	Display counters of all axes & select the type of the counters.
	Offset	Set & display tool offset data.
	Coord	Set & display workpiece coordinate system offset.
	Cnt set	Set the relative position counter to an arbitrary value.
	MST	Set & display each command for S, M, T & 2nd M functions.
	Modal	Display the execution modal value of the machining program.
	Tree	Display program/MDI interrupt/user macro call in nesting structure.
	Time	Set & display integrating time (date, time, power ON time, etc.).
	Com var	Set & display the details of common variables.
	Loc var	Specify a nesting level of subprogram & display local variables.
	P corr	Correct & change the next command by block stop during auto-/MDI op.
	PLC SW	Switch ON/OFF each NC ops. control signal allocated by ladder program.
	G92 set	Perform origin set & origin cancel.
	Col stp	Register arbitrary collation & stop position set as single block stop.
	LD MTR	Spindle load and Z axis load, etc. can be displayed in meter.
	Sp-stby	The spindle tool No. and the standby tool No. can be displayed.
	TipDisp	Display tool tip pos, handle interrupt amount and tool tip speed.
	All sp	Display commanded/actual rot speed values of all the spindles.
Setup	T-ofs	Set & display tool offset data.
	T-meas	Measure T length & radius manually & set them as tool offset amounts.
	T-reg	Register T No. according to the magazine pot, spindle & wait position.
	T-life	Scale workpiece to figure face/hole ctr/wid ctr & set as coord ofs.
	Coord	Set & display workpiece coordinate system offset.
	W-meas	Set & display user parameters.
	User	Display & edit MDI programs set in NC memory.
	MDI	Edit (add/delete/change) programs in NC memory & create new ones.
	Cnt set	Set the relative position counter to an arbitrary value.
	MST	Set & display each command for S, M, T & 2nd M functions.
	T-list	Reference & display T-codelist.
	Pallet	The machining program is registered into the pallet of APC.
Edit	Edit	Edit (add/delete/change) programs in NC memory & create new ones.
	Check	Trace the T path based on the machining program w/o running machine.
	NAVI	Create the part program simply.
	I/O	Input/Output machining programs betw. NC memory & external I/O device.

Screen	Menu name	Outline
Diagn	Config	Display the H/W & S/W (S/W No. & version) configurations.
	Option	Display the details of the options registered in NC memory.
	I/F dia	Set & display input/output signals of the ladder program.
	Drv mon	Display drive diagnostic information. (servo/spindle/power supply).
	Mem dia	Set & display NC internal data.
	Alarm	Display a list of currently occurring alarms & their messages.
	Selfdia	Display the H/W & Operation stop state.
	NC Smp	Set Sampling parameter & Sample NC internal data.
	A net	Customer service is available by connecting NC with NC service.
	MTB net	Machine maker service is available by connecting NC & machine maker.
Mainte	Mainte	Perform NC memory format/abs. para. setting/maint. data backup, etc.
	Param	Select a type of parameter to set & display the parameter values.
	I/O	Input/Output machining programs betw. NC memory & external I/O device.

1.7.1 Displaying the Menu Function Outline

Operation method

- (1) Press the **MenuList** key.  The Menu list window appears showing the selected screen's menu list at the top.
- When the **MenuList** key is pressed while editing the file on the Edit screen, a prompt to confirm whether to save the program appears before Menu list window opens.
- (2) Using the , , ,  and page keys, move the cursor to the menu for which the function outline is to be displayed.  The function outline for the selected menu appears. The function outline does not appear if a menu with no name is selected.
- The cursor does not move to the group name.

1.7.2 Directly Moving to the Menu Function

Operation method

- (1) Press the **MenuList** key.  The Menu list window appears showing the selected screen's menu list at the top.
- (2) Using the , , ,  and page keys, move the cursor to the menu to move to.  The function outline for the selected menu appears.
- (3) Press the **INPUT** key.  The selected menu function can be moved to directly. The menu is not moved to if a menu with no name is selected. The Menu list window closes after moving.

1.7.3 Moving the cursor

Operation method

(1) Press the **MenuList** key.



The Menu list window appears showing the selected screen's menu list at the top.



(2) Press the **↓** key four times.



The cursor moves down four times, and the screen scrolls.



(3) Press the **↑** key three times.



The cursor moves up three times, and the screen scrolls.



(4) Press the **PageDown** key once.



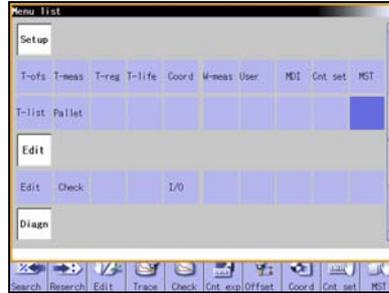
The screen scrolls.



(5) Press the  key once. **(Note 1)**



The cursor moves to the upper right, and the screen scrolls.



(6) Press the  key once.



The screen scrolls.



(7) Press the  key twice.



The cursor moves down twice.



(Note 1) The cursor moves in the same way even when the  key is pressed.

(Note 2) The cursor moves in the same way even when the  key is pressed.

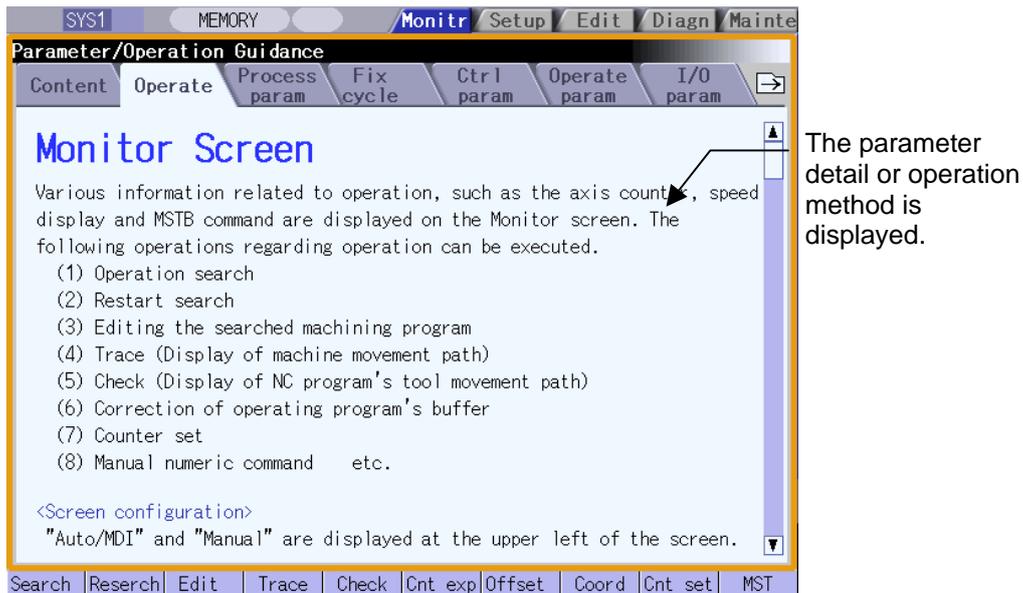
1.8 Guidance Function

1.8.1 Parameter Guidance

The parameter/operation guidance function displays the details of the parameters or the operation methods according to the state of the screen currently being displayed.

Screen configuration

The parameter/operation guidance window is displayed with the following type of configuration.



Operation method

If the **[?]** key is pressed on any screen, the guidance window will open. If a pop-up window other than the guidance window is open, the guidance window will open over the currently opened pop-up window. In this case, the menu state does not change. If **[?]** key or **CANCEL** key is pressed again when the guidance window is open, the guidance window will close and the screen will return to the previous state in which the **[?]** key was pressed.

Displaying the operation guidance

(1) Press the **Edit** menu on operation screen.



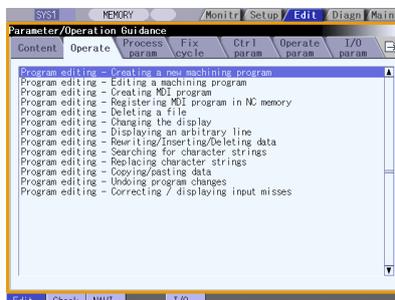
The edit window is displayed.



(2) Press the **?** key.



The guidance window is opened, and sub-contents for edit operation method is displayed.

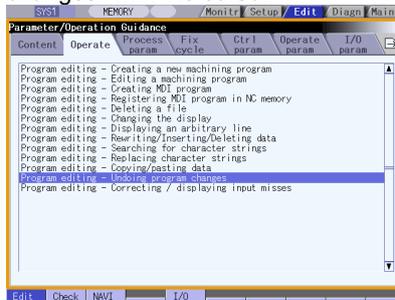


(Note) If the explanation of operation method is nothing, the contents list is displayed.

(3) Move the cursor to the operation method to be displayed by **↑**, **↓**, **▲**, **▼** keys.



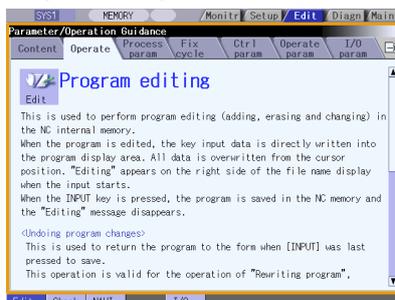
Select the "Program editing - Undoing program changes" with the cursor.



(4) Press the **INPUT** key.



The operation method for "Program editing - Undoing program changes" is displayed.



(Note 1) If the parameter tag such as machine parameter, fixed cycle, etc. is opened with the **Ⓜ** key, the details for the top parameter are displayed.

Displaying the parameter guidance

- (1) Press the **Param** menu on maintenance screen. → The machining parameter screen is displayed.

- (2) Press the **↓** key, and move the cursor to "#8005 ZONE r" parameter. →

No.	Name	Data	No.	Name	Data
<MRK COUNT>			<FIXED C.>		
8001MRK COUNT M		30	8012/673 n		0.000
8002MRK COUNT		44	8013/683 n		0.000
8003MRK COUNT LIMIT		333	8014/COZ-VALE		0
<AUTO TLM>			8015/COZ-ANGLE		0
8004SPEED		0	8016/671 MINIMUM		0.000
8005 ZONE r		0.000	8017/671 DELTA-D		0.000
8006 ZONE d		0.000	8018/684/674 n		0.000
<AUTO CORNER OVR.>			<PRECISION>		
8007/OVERRIDE		0	8019/R COMP		0
8008/MAX ANGLE		0	Theor. R decrease		2.296
8009/DSC_ZONE		0.000	8020/DCC_ANGLE		0
<T-TIP OFFSET>			8021/COMP_CHANGE		0
8010/ABS.MAX		0.000	8022/CORNER COMP		0
8011/LNC.MAX		0.000	8023/CURVE COMP		0

- (3) Press the **?** key. →

- The guidance window is opened, and the detail of "#8005 ZONE r" parameter is displayed.

#8005 ZONE r

Contents
Set the distance between the measurement point and deceleration start point.

Setting range
0 to 99999.999 (mm)

- (4) Press the **↩** key. →

- The operation method for parameter screen is displayed.

Parameter screens

The various parameters can be displayed and set on this screen.

<Changing the parameter display>

Changing to the computer link parameters

- 1) Press the menu change key until the menu [Link param] appears.
- 2) Press the menu [Link param].

<Selecting the parameter No.>

Selecting the parameter No.8071

- 1) Press the menu [Param No.]

(Note 1) While guidance window is displayed, the parameter details displayed on each parameter type tab are recorded. Therefore, when the parameter type is switched with **↩**/**➡** key, last displayed parameter details are displayed.

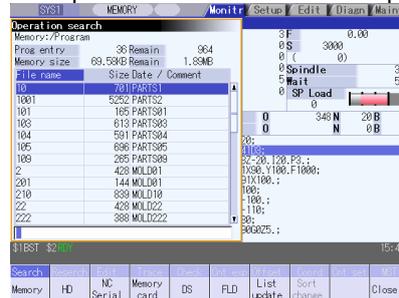
(Note 2) If the parameter No. cannot be gotten (the cursor is non-display, the cursor is put on blank line, the cursor is put on comment line), the top of parameter guidance for each parameter appears.

Displaying the operation guidance by selecting the contents

- (1) Press the **Search** menu on operation screen.



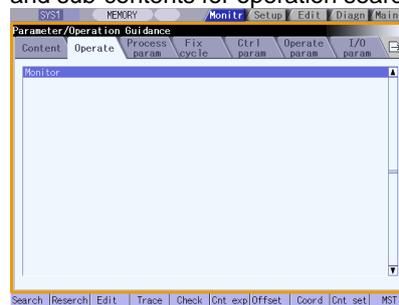
The operation search window is displayed.



- (2) Press the **?** key.



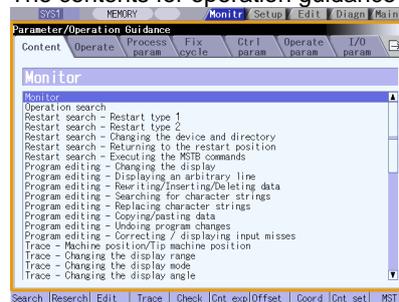
The parameter/operation guidance window is opened, and sub-contents for operation search is displayed.



- (3) Press the **Enter** key.



The contents for operation guidance are displayed.



- (4) Move the cursor to the operation method to be displayed by **↑**, **↓**, **▲**, **▼** keys.



Select the "Restart search - Restart type 2" with the cursor.



(5) Press the **INPUT** key.



The operation method for "Restart search" is displayed.



(Note 1) The operation details of function that cannot be selected because of no option are also displayed on the contents list.

(Note 2) When the  key is pressed during the contents display, the sub-contents for operation search (operation method (2)) are displayed.

Precaution

- (1) If the  key is pressed when alarm occurs, the alarm guidance window will open.
- (2) To display another operation method, select newly an operation method from the contents list. It is not possible to jump from the parameter, operation details to other details.
- (3) When the  key is pressed while the menu list window is displayed, the guidance window is opened after the menu list window is closed.

1.8.2 Alarm Guidance

The alarm guidance is the function that displays message, details and remedy for the currently occurring alarms.

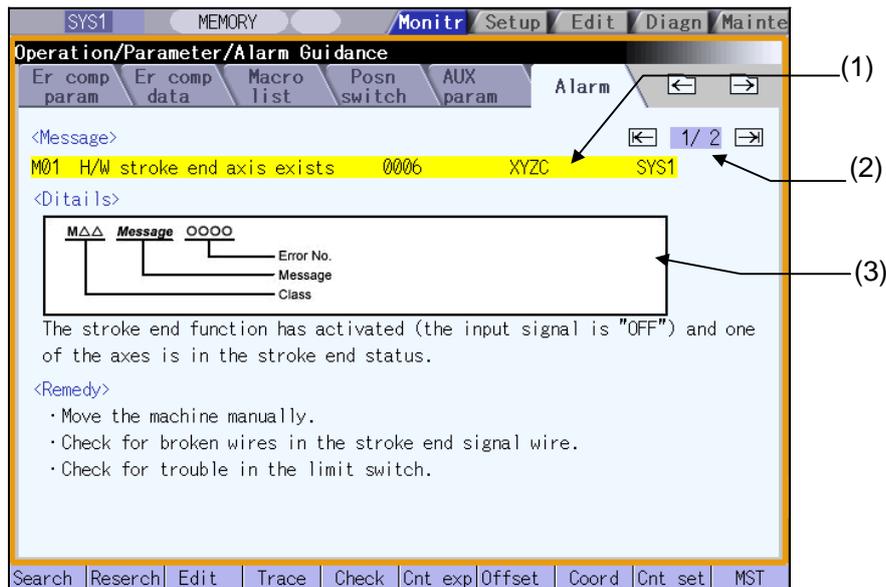
This function is an option.

Screen configuration

The alarm guidance is displayed in "alarm" tag on "operation/parameter/alarm guidance window".

The "alarm" tag is on the rightmost end of the guidance window.

A scroll bar appears when details and remedy, etc. do not fit in one page.



Display items

Display item	Details
(1) Message	This displays "NC alarm" and "PLC alarm message". The displayed content is same as the content displayed in the alarm message of the diagnosis screen.
(2) Number of page	This displays "order of priority / total number of occurring alarms".
(3) Details/Remedy, etc.	This displays detail and remedy, etc. for alarm message.

Operation method

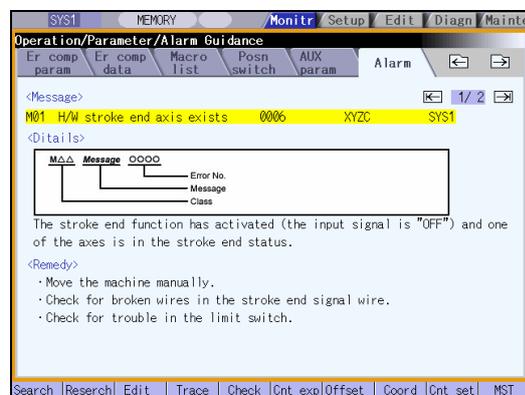
If the  key is pressed on any screen, the alarm guidance window will open. If a pop-up window other than the alarm guidance window is open, the alarm guidance window will open over the currently opened pop-up window. In this case, the menu state does not change. If  key or **CANCEL** key is pressed again when the guidance window is open, the guidance window will close and the screen will return to the previous state in which the  key was pressed.

Displaying the alarm guidance

- (1) Press the  key during occurring alarm on operation screen.
(Ex.) When "H/W stroke end axis exists" and another alarm occur.



The guidance window is opened while "alarm" tag is valid. The alarm details and remedy are displayed for "H/W stroke end axis exists".



- (Note 1)** The alarm guidance is executed for the alarm that is occurred when the  key has been pressed. Therefore, the guidance will be displayed continually even if the alarm is reset while the alarm guidance is displayed. The alarm guidance is not executed for the alarm that has occurred while the alarm guidance is displayed.
- (Note 2)** If the  key is pressed when an alarm is not occurring, the operation/parameter guidance appears. In this state, if the "alarm" tag is selected with the right frame key, nothing is displayed in the message, details and remedy columns. (These columns are blank.)
- (Note 3)** All the explanations of the alarm having the same error class and No. are displayed in the details column.

Changing the alarm guidance

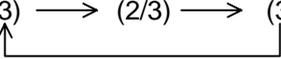
- (1) Press the  key or  key when the alarm guidance is displayed.



The alarm guidance with one priority lower than the current alarm guidance is displayed.

If the  key or  key is pressed when the lowest priority alarm is displayed, the alarm with the highest priority is displayed.

(Ex.) When 3 alarms occur.

(1/3)  (2/3)  (3/3)


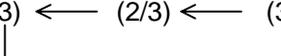
- (2) Press the  key or  key when the alarm guidance is displayed.



The alarm guidance with one priority higher than the current alarm guidance is displayed.

If the  key or  key is pressed when the highest priority alarm is displayed, the alarm with the lowest priority is displayed.

(Ex.) When 3 alarms occur.

(1/3)  (2/3)  (3/3)


(Note 1) Up to 10 alarm guidance displays are displayed in order of priority.

(Note 2) Even if the , ,  or  key is pressed when only one alarm occurs, the screen display is not switched.

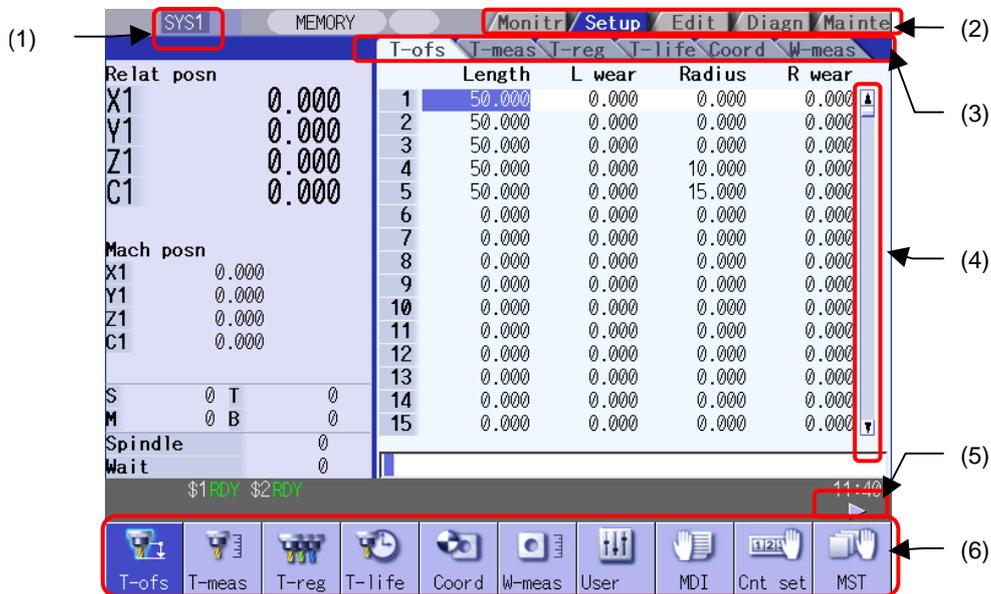
Precautions

- (1) While the user parameter window on "Setup" or the parameter screen on "Mainte" is displayed, the parameter guidance display will be prioritized even if an alarm occurs.
- (2) When the  key is pressed while the menu list window is displayed, the guidance window is opened after the menu list window is closed.
- (3) The alarm guidance of stop code and operator message is not displayed.

1.9 Touch Panel Functions

Touch panel screen operation is available with a touch panel display unit.

1.9.1 Basic Operation



Display items

Display item	Details
(1) System	This switches from the current part system to the next current system.
(2) Main tag	A selected screen will open. As in the case with a screen change by key inputting, if a popup window is displayed, close it before changing screens. If there is a file under editing, save the file before changing screens.
(3) Sub tag	This displays the contents of the selected tag. When a tag is changed while operation menu is on a wait to be input, the input wait status is cancelled and then the tag is changed. Sub tags are available on Setup screen, Edit screen, parameter/operation guidance window.
(4) Scroll bar	If [▲][▼] of a scroll bar is touched, the view is scrolled one line up/down. If any part of scroll bar besides tabs is touched, the page jumps to the next/previous one. (Note) In some screens, if [▲][▼] is touched, the page jumps to the next/previous one, and scrolling by line is not possible.
(5) Next menu	This switches to the next menu if available.
(6) Menu	When a menu is selected by touching, the menu's process is executed.

Moving cursor by touch panel

When a place within the data setting area of the screen having a cursor is touched, the cursor moves to the touched position. Cursor movement by touch panel operation is also possible in the area where the cursor is moved by , , , , keys. When a window is closed after cursor movement, the cursor position is returned to the first line. When the cursor is moved or the area is changed by touch panel operation in the Y/N input wait status (when the [Line clear] menu is selected, etc.) the status will be that when N is selected (canceled), and then the cursor movement or area change will be carried out.

(Example) Cursor movement while waiting for Y/N input

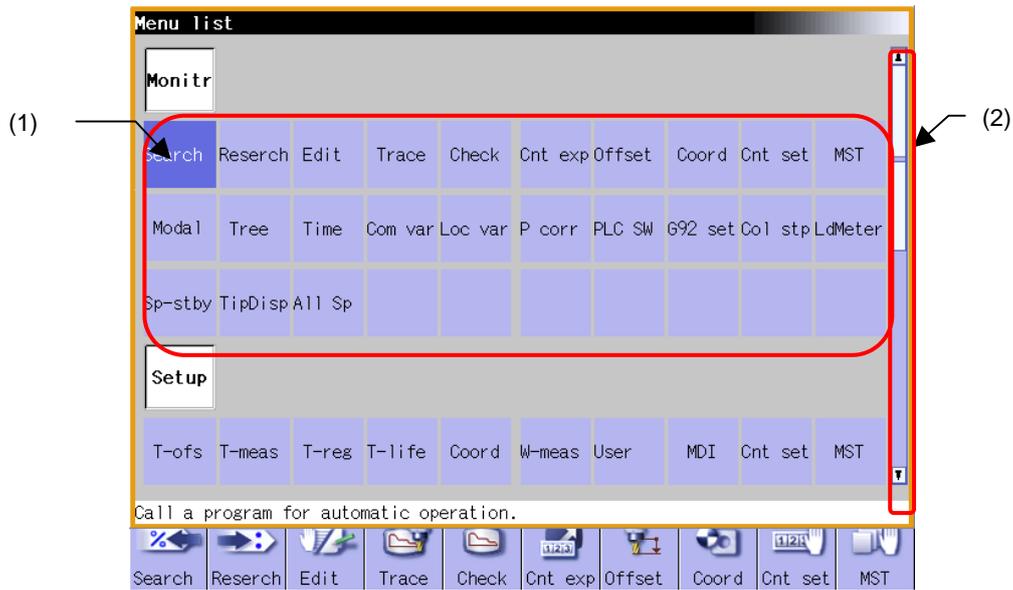
- (1) Select the **Coord** menu on the monitor screen. → Workpiece coordinate system offset window is displayed.
- (2) Select the **ALL clear** menu. → The **ALL clear** menu is highlighted. The message "Clear all?(Y/N)" is displayed.
- (3) Touch the coordinate value for the arbitrary coordinate system offset. → The cursor is moved to the touched position. The highlight of the **ALL clear** menu is undone and "All clear" is canceled. The message "Clear all?(Y/N)" disappears.



Changing active area

In the screen where active area can be changed, the area touched will become active. Active area cannot be changed by touch panel operation during mass-editing or file saving for automatic backup.

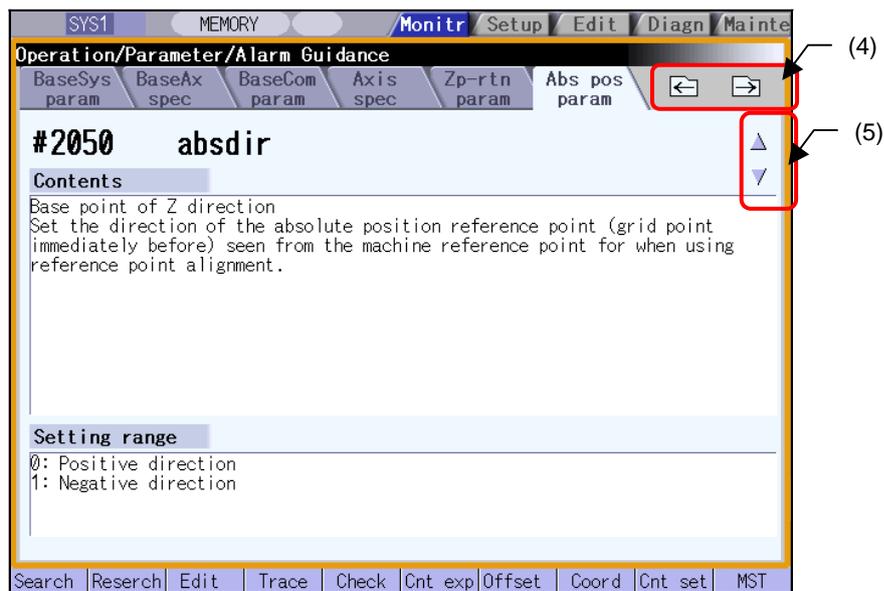
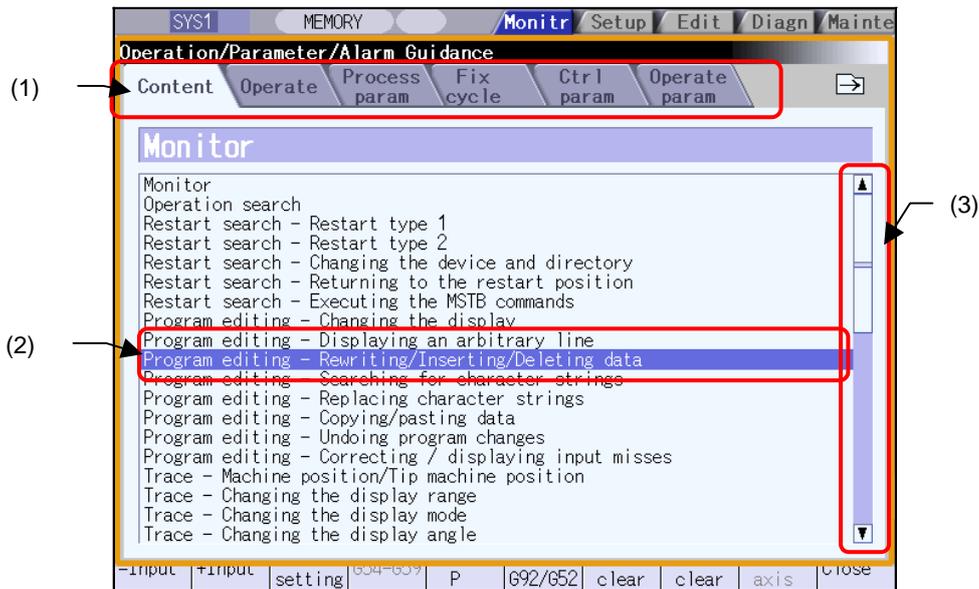
1.9.2 Menu List



Display items

Display item	Details
(1) Menu selection	The menu touched is turned active. Its screen will be displayed by touching that active menu once again or by pressing the INPUT key.
(2) Scroll bar	If [▲][▼] of a scroll bar is touched, the view is scrolled one line up/down. If any part of scroll bar besides tabs is touched, the page jumps to the next/previous one.

1.9.3 Operation/Parameter Guidance



Display items

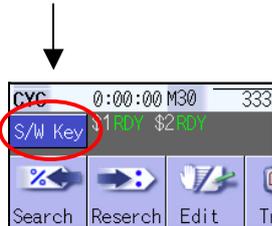
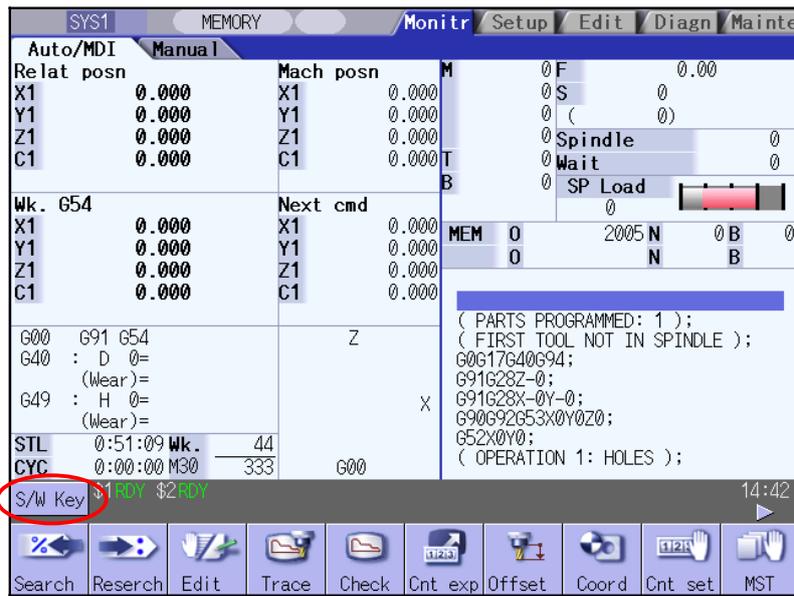
Display item	Details
(1) Guidance tag	Type of guidance is changed.
(2) Cursor	When either "Content" tag or "Operate" tag is selected, if an item is touched, the cursor moves to that item. The contents will be displayed by touching the item at the cursor position once again or by pressing the INPUT key.
(3) Scroll bar	If [▲][▼] of a scroll bar is touched, the view is scrolled one line up/down. If any part of scroll bar besides tabs is touched, the page jumps to the next/previous one.
(4) Tag change	Active tab shifts one toward right/left.
(5) Page change	By touching ▲ mark, previous page is displayed; by touching ▼ mark, next page is displayed.

1.10 Touch Panel S/W Key

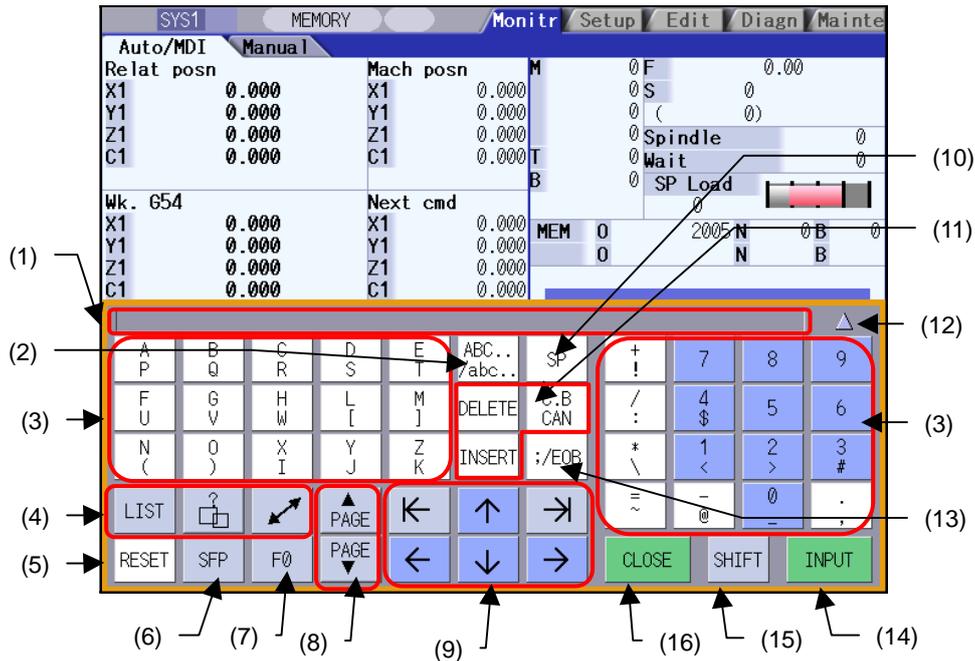
When a touch panel display unit is used, touch panel operation is available.

With the touch panel soft key function, a key input is possible by touching a key on the S/W keyboard which is displayed on the screen. S/W keyboard is displayed if **S/W Key** button is touched in each screen. In this case, screen or menu status will not change. S/W keyboard is always displayed on the front screen. When the CLOSE key on the S/W keyboard is touched, the S/W keyboard will be closed. S/W keyboard will be enabled when the parameter "#11010 Software keyboard" is set to "1". Even if the parameter is 0, S/W keyboard can be used when the keyboard has not been connected. When S/W keyboard is disabled, **S/W Key** button is not displayed.

A horizontal S/W keyboard is displayed in 700 series, and a vertical S/W keyboard is displayed in 70 series,



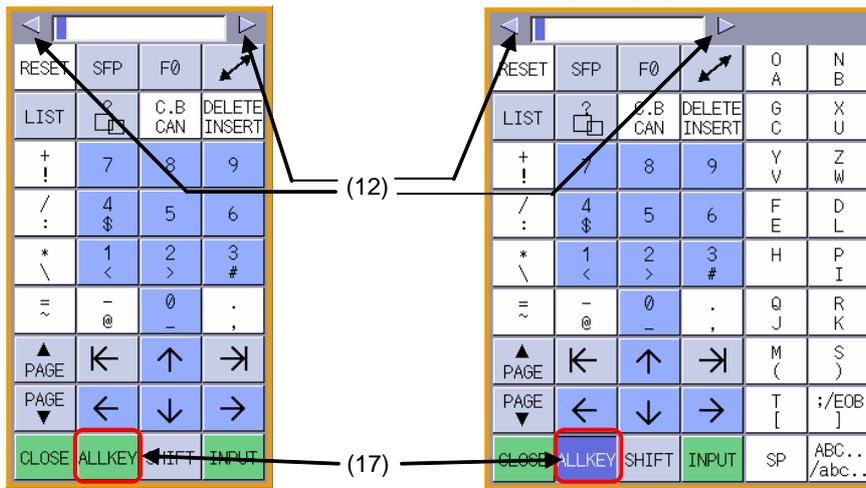
<700 Series>



<70 Series>

• Numerical keypad display

• All keys display



Display items

Display item	Details
(1) Input area	This displays a character string according to the keys touched by the operator.
(2) Lower-case letter input key	This changes upper/lower-case of alphabet. The ABC../abc.. key is highlighted when the lower-case-letter-setting is valid.
(3) Data setting key	This enters characters in the input area or screen.
(4) Window operation key	LIST : This displays the menu list window. ? : This displays the parameter/operation guidance window.  : This displays the window.  : This changes active windows.
(5) Reset key	This resets the NC.
(6) SFP key	This displays the onboard screen or custom screen.

Display item	Details
(7) F0 key	This displays the onboard screen.
(8) Page change key	This displays the contents in the previous/next page.
(9) Cursor key	If data is set in the display items on the screen, the cursor is moved to up, down, left and right.
(10) Blank key	This enters a blank.
(11) Data correction key	<p>INSERT : The mode changes to data insertion mode, and characters are input at the current cursor position when data setting key is pressed. When DELETE, C.B CAN, INPUT, cursor key, or tab key, etc. is pressed, or when the screen is changed to another, the mode returns to overwriting mode.</p> <p>DELETE : This deletes a letter before the cursor position.</p> <p>C.B : This cancels the settings in the input area.</p> <p>CAN : This cancels.</p>
(12) Display position switch	This switches the display position of the S/W keyboard.
(13) Block end key	This enters ";".
(14) INPUT key	This confirms the data entered in the input area or screen.
(15) Shift key	This validates the setting on the lower line of data setting key. Once SHIFT key is selected, it will be valid until another key is pressed or the S/W keyboard is closed. Menus are displayed in highlight while SHIFT key is valid.
(16) CLOSE key	This closes the S/W keyboard.
(17) Key display switch key	This switches numerical keypad display/all keys display.

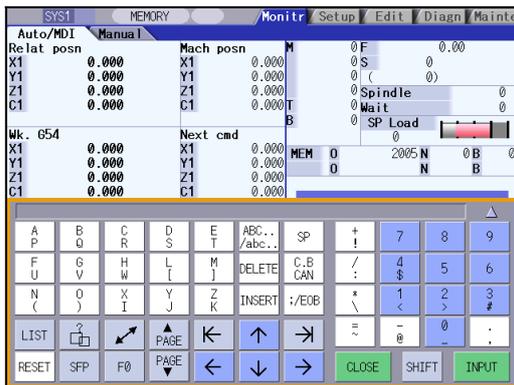
1.10.1 Operation Method

Displaying/closing the S/W keyboard

- (1) Touch the **S/W Key** button. → S/W keyboard is displayed on the front screen. **S/W Key** button is highlighted.
- (2) Touch the **CLOSE** key, or touch the **S/W Key** button again. → S/W keyboard will close.

Changing the display position

- (1) Touch the **S/W Key** button. → S/W keyboard is displayed on the front screen. **S/W Key** button is highlighted.
- (2) Touch **△** located at the upper right corner of the S/W keyboard. → S/W keyboard is displayed on the upper side of the screen. **△** changes to **▽**.



- (3) Touch **▽** located at the upper right corner of the S/W keyboard. → S/W keyboard is displayed on the upper side of the screen. **▽** changes to **△**.

(Note 1) In 70 series, the keyboard can be moved to the right/left by touching **◁ ▷** on the upper part of the S/W keyboard.

Entering data when the input area is enabled

If displaying the S/W keyboard while input area is being displayed on a screen, such as in the case where the operation search window or setup screen is being displayed, the S/W keyboard input area will be valid. At this time, the key selected on the S/W keyboard is temporarily displayed in the S/W keyboard input area, and then is input and set in the screen by selecting the **INPUT** key.

(Example) Set the value in T tool compensation amount

- (1) Display T compensation tag on the setup screen and touch the **S/W Key** button. → S/W keyboard is displayed.
S/W keyboard input area will be invalid.
- (2) Touch the , , ,  keys to move the cursor to an arbitrary position. → The cursor for tool compensation amount is moved.
- (3) Touch the data setting key and enter the value.
Example: 15 → "15" is displayed in the S/W keyboard input area.
- (4) Touch the **INPUT** key. → "15" is input at the tool compensation amount cursor position.

Entering data when the input area is disabled

If S/W keyboard is displayed while edit window is displayed for program editing or input area is not displayed on a screen, or when the monitor screen main menu is displayed, S/W keyboard input area will be invalid. At this time, the key selected on the S/W keyboard is directly input to the screen.

- (1) Display the edit window on the monitor screen and touch the **S/W Key** button. → S/W keyboard is displayed.
S/W keyboard input area will be invalid.
- (2) Touch the , , ,  keys to move the cursor to an arbitrary position. → The cursor for the edit window is moved.
- (3) Touch the data setting key and enter the value.
Example: G28 → "G28" is displayed in the edit window.
- (4) Touch the **INPUT** key. → "G28" which has been input in the edit window is confirmed.

1.10.2 Automatic Display of S/W Keyboard [70 Series Only]

When "2" is set to parameter "#11010 Software keyboard", the S/W keyboard is displayed automatically in 70 series when a specific screen is displayed.

The S/W keyboard is displayed automatically on the following screens.

Screen (S/W keyboard is displayed automatically)		Remarks
Monitor screen	Operation search	
	Restart search	
	Program edit	
	Tool compensation	
	Workpiece coordinate system offset	
	Counter set	
	Origin set	
	Manual number value command	
	Integrated time display	
	Common variables	
	Buffer correction	
	Verify stop	
Setup screen	User parameter	
	MDI program editing	
	Counter setting	
	Manual numerical value command	
	T code list	
Edit screen	Program editing	Displayed when the menu Open , Open (New) or Erase file is pressed.
	Program check	Displayed when the menu Check search is pressed.
	Program input/output	Displayed when the menu is pressed.
Maintenance	Parameter	
	Input/output	

1.11 Screen Saver (Backlight OFF) Function

The screen saver function protects the display unit by backlight OFF after the time set in the parameters has elapsed.

The backlight can also be turned OFF with key operations on the Monitor screen.

The backlight is turned ON by pressing any key, or by touching anywhere on the screen when the display unit carries a touch-sensitive screen.

1.11.1 Backlight OFF

The screen saver function has two types: "Automatic change function" that the backlight is turned OFF after a certain period of time, and "Manual change function" that the backlight is turned OFF by key operations.

Automatic change function

If there is no key input, no operation from touch-sensitive screen, or no screen display request signal input from the machine within the time set in the parameter "#8078 Screen Saver Time", the backlight will be turned OFF.

If "#8078 Screen Saver Time" is set to "0", the backlight will not be turned OFF.

Manual change function

The backlight can be turned OFF by pressing the **SHIFT** and **C.B CAN** keys on the Monitor screen (when a pop-up screen is not displayed). It is possible to be turned OFF by inputting the keys from the S/W keyboard. Even if "#8078 Screen Saver Time" is set to "0", the backlight can be turned OFF by pressing the **SHIFT** and **C.B CAN** keys.

(Note 1) Even if the **SHIFT** and **C.B CAN** keys are pressed when a screen other than the main menu is displayed on the Monitor screen (when a pop-up window is displayed), the backlight cannot be turned OFF.

(Note 2) The backlight cannot be turned OFF by the manual change function on screen other than Monitor screen.

1.11.2 Backlight ON

If a key is pressed or the screen display request signal is input while the backlight is OFF, the backlight will be turned ON.

The backlight is turned ON by touching anywhere on the screen if the display unit carries a touch-sensitive screen. In that case, even if the screen name tag or the menu is touched, the display is not changed.

Key operation

The correspondence when the key is pressed is as follows during backlight OFF.

(1) Backlight ON + screen transition

(Example) Function key, Part system changeover key, Frame keys,  key, etc.

(2) Backlight ON only

(Example) Page changeover key, Menu changeover key, Data setting keys, Data correction keys, etc.

(3) The key input is ignored

(Example)  key,  key,  key,  key

Precautions

- (1) The backlight will not be turned ON even if the reset key is pressed. However, if the screen display request signal is input when the reset key is pressed, the backlight will be turned ON. Refer to the instruction manual issued by each machine tool builder for details. Note that whether the backlight will be turned ON when any of the machine operation board keys (other than the NC operation board keys) is pressed will differ according to the machine specifications. Refer to the instruction manual issued by each machine tool builder for details.
- (2) If a key is pressed or the screen display request signal is input while the screen is displayed, counting of the time to turn the backlight OFF will restart. However, even if ,  or  key is pressed, counting of the time to turn the backlight OFF will not restart.
- (3) Operation when the  or  key is pressed will differ according to the machine tool builder. Refer to the instruction manual issued by each machine tool builder for details.
- (4) When the backlight is turned ON, the first character is ignored if the  +  to  key is input. (Ex. WHILE -> HILE)
- (5) The S/W keyboard is operated regardless of backlight ON/OFF. However, when the backlight is OFF, the key input is ignored even if the following keys are input from S/W keyboard.
 - Data setting keys
 - Data correction keys
 - SP key
 -  key
 - Cursor keys
 - Page changeover key

1.12 Screen Capture [70 Series Only]

When "1" is set to the parameter "#8121 Screen Capture", and then **SHIFT** key is kept pressing for about 3 seconds until the buzzer sounds the screen information displayed in the display unit can be output to memory card (CF) as a file of the bitmap format.

Operation method

- (1) Set the parameter "#8121 Screen Capture" to "1".
- (2) Display the acquired screen.
- (3) Hold down the **SHIFT** key on keyboard for three seconds until the buzzer sounds.  When the **SHIFT** key is pressed, the buzzer sounds "blip". About three seconds later, the bitmap file output is completed when the buzzer sounds "blip" again.
- (4) Return the parameter "#8121 Screen Capture" to "0".

Bitmap file format

The bitmap file format output by the screen-capture function is as follows.

Color number	Color (256 colors)
Size	308,278 byte (640 x 480)
File name	A day of the week_Month_Day_Hour_Minute_Second_Year.bmp (Ex.) When output at 2007/1/24 (Wed) 12:18:38 WED_JAN_24_12_18_38_2007.bmp

Precautions

- (1) The static image only can be output.
- (2) The bitmap file cannot be output during automatic start.
- (3) Do not output the bitmap file while the axis is moving by the manual operation mode etc.
- (4) The screen cannot be operated while the bitmap file is being output to memory card (CF). Do not turn OFF the power supply while being output. The memory card may be damaged.
- (5) The bitmap file cannot be output when the memory card is not inserted.
- (6) When there is no empty capacity in the memory card, the bitmap file cannot be correctly displayed.
- (7) The number of bitmap files which can be stored in the memory card is nine files or less.

1.13 Multi-part System Program Management

1.13.1 Outline

Separate programs, used in each part system, can be managed under a common name in the multi-part system. This function facilitates the management of the process programs that are simultaneously executed in the multi-part systems.

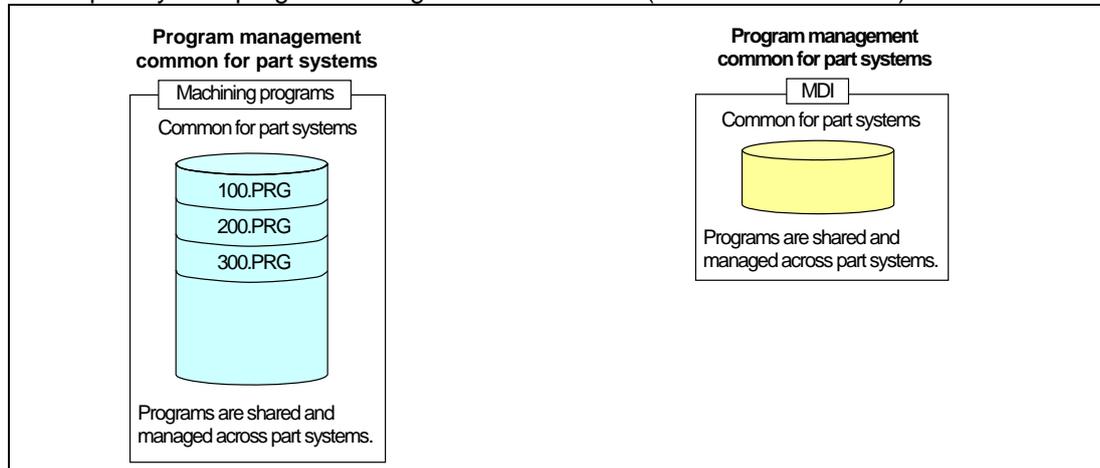
The multi-part system program management is turned valid/invalid with a parameter "#1285 ext21/bit0 Multi-part system program management".

The followings are available when the multi-part system program management is valid:

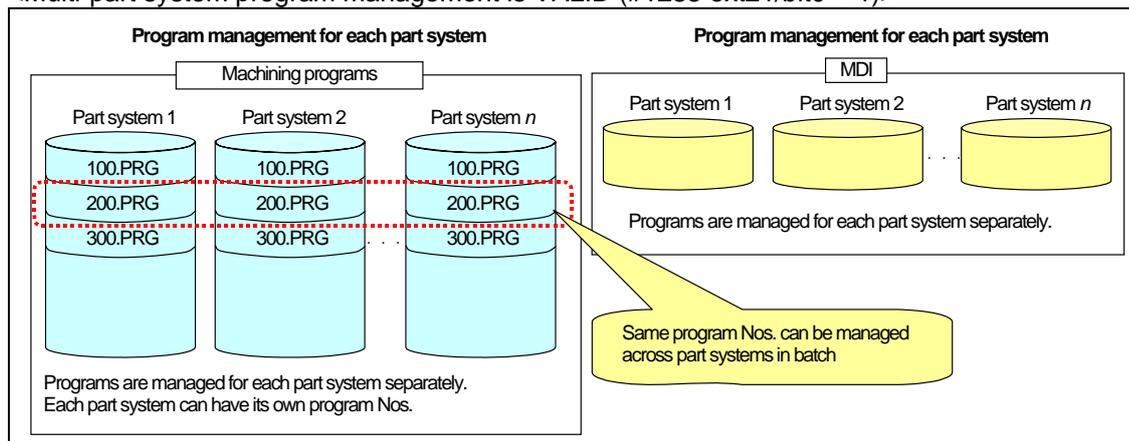
- Executing the program search across all part systems in batch. **(Note)**
- Saving/editing the programs for each part system under one program name.
- Inputting/outputting programs of all part systems in batch.

(Note) While the multi-part system program management is valid, there is an option whether the operation search, check search or restart search is executed across all part systems in batch or is executed for each part system separately. Change the search method with the parameter "#1285 ext21/bit1 Program search method changeover".

<Multi-part system program management is INVALID (#1285 ext21/bit0 = 0)>



<Multi-part system program management is VALID (#1285 ext21/bit0 = 1)>



1.13.2 Restrictions

- (1) This function is provided for NC memory (including MDI and machine tool builder macro) only.
- (2) A file name can have up to 29 characters, including the extension. (Up to 32 characters when this function is invalid.)
- (3) The tape mode does not allow the batch search across all part systems.
- (4) Set a verification stop position in a part system separately, even if this function is ON. Another verification search position is required for the other part system.
- (5) The restart search can be executed for all part systems in batch when type 1 is selected and no ONBP No. is specified.
- (6) The program check (2D)/(3D) allows the check search for all part systems in batch, although the drawings are not common for all part systems. The Check continu and Check step menus are operated for the displayed part system only.
- (7) Files are not created separately for each part system when the machining programs are output to external devices on the input/output screen. Machining programs of all part systems are combined to one file before being output.

2. Monitor Screens

Various information related to operation, such as the axis counter, speed display and MSTB command are displayed on the Monitor screen. The following operations regarding operation can be executed.

- (1) Operation search
- (2) Restart search
- (3) Editing the searched machining program
- (4) Trace (Display of machine movement path)
- (5) Check (Display of NC program's tool movement path)
- (6) Correction of operating program's buffer
- (7) Counter set
- (8) Manual numeric command etc.

2.1 Screen Configuration

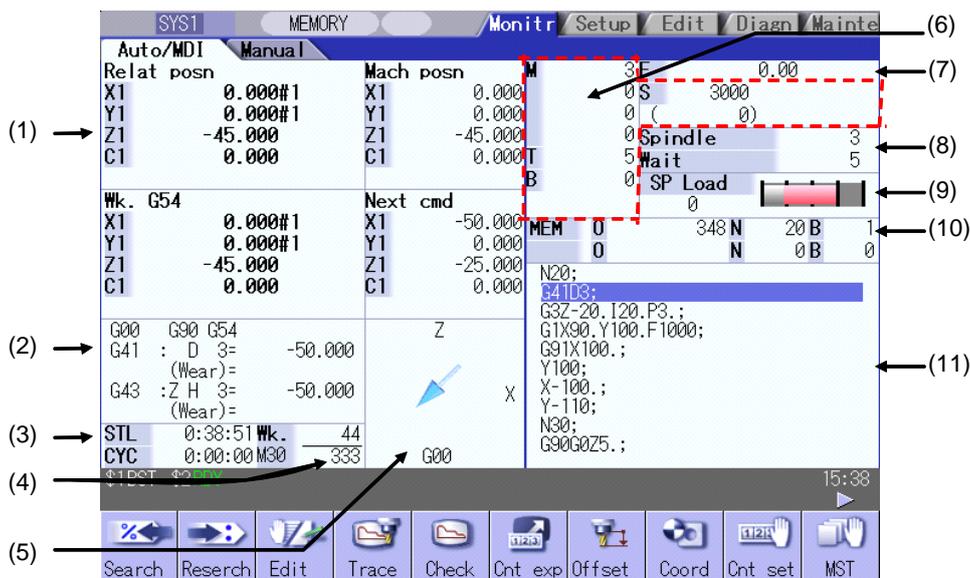
As for the screen layout, the content of the display is different depending on whether to enable the 2-part system simultaneous display (#11019 2-system display), operation mode, and the number of counter display part systems as shown in the table below.

2-part system simultaneous display (#11019 2-system display)	Operation mode	Number of counter display part systems
Disable	Automatic	5 axes or less 6 axes or more
	Manual	-
Enable	Automatic/ Manual	4 axes or less 5 axes or more

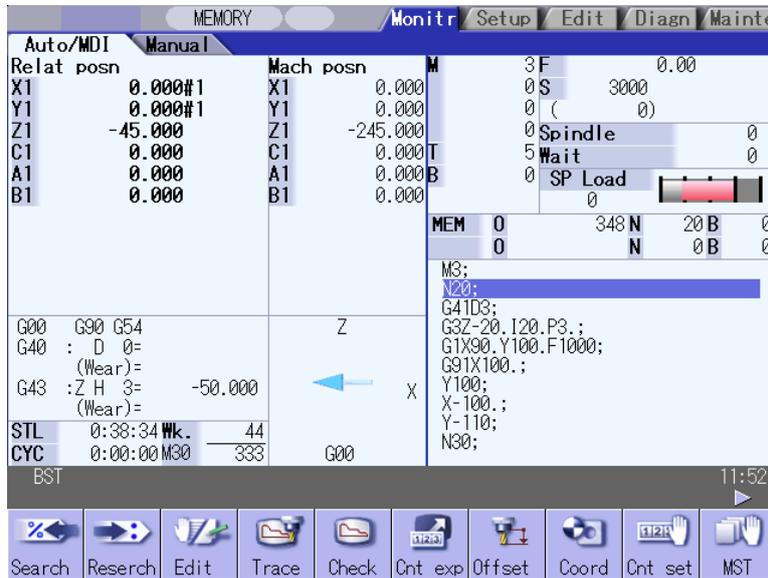
[1-part system display] ... The parameter "11019 2-system display" = "0"

"Auto/MDI" and "Manual" are displayed at the upper left of the screen. These displays change according to the mode selection switch. The details displayed for [Auto/MDI] differ according to the number of NC axes.

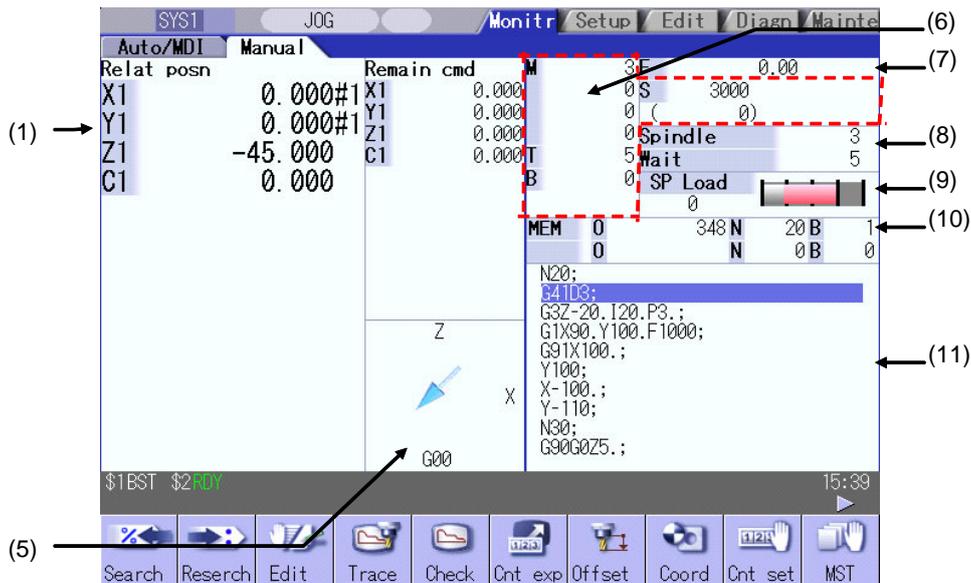
<For Auto/MDI (Part system with five or less axes)>



<For Auto/MDI (Part system with six or more axes)>



<For Manual>

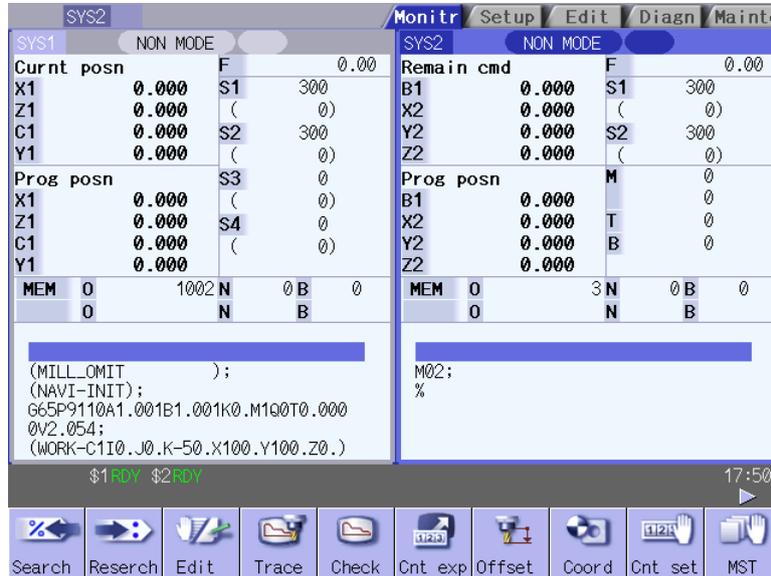


[2-part system simultaneous display] The parameter "#11019 2-system display" = "1"

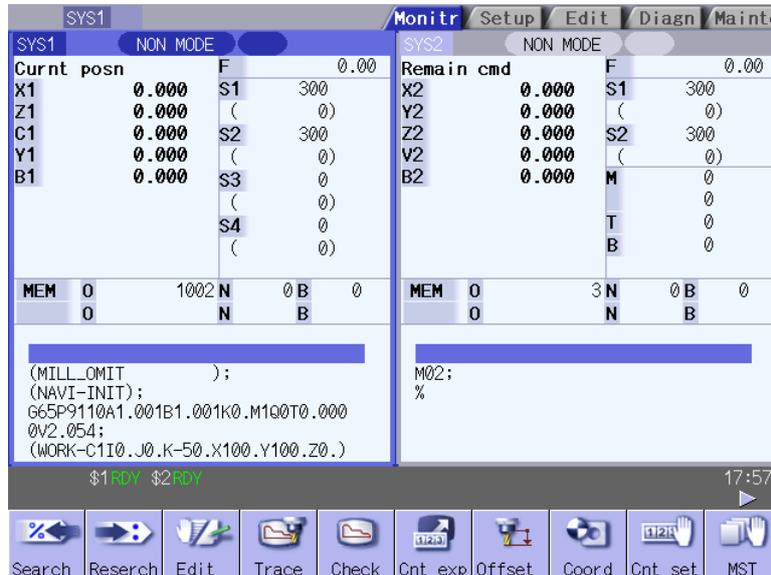
The screen layout is the same at automatic operation and manual operation.

However, the content of the display is different depending on the specifications of the number of NC axes.

<The counter displayed axes are four or less>



<The counter displayed axes are five or more>



Display items

Display item	Details
(1) Counter display	<p>This displays the counter of the relative position and workpiece coordinates positions, etc.</p> <p>If each axis is in a specific position or status, the following status symbol appears.</p> <p>#1 to #4 : No. 1 to No. 4 reference position : Servo OFF state MR : Mirror image >< : Axis removed state CT : Auxiliary axis state</p> <p>Whether to allow for the tool length compensation and tool radius compensation can be set with the parameter "#1287 ext23/bit4".</p> <p>Whether to display the axis counter can be set by the parameter "#1069 no_dsp".</p>
(2) G modal simple display (1-part system display only)	<p>This displays the modal status.</p> <p>G command modal status of Group 1 G command modal status of Group 3 Selected workpiece coordinate system Tool radius compensation modal, compensation No. shape compensation amount, tool radius wear amount Tool length compensation modal, compensation axis name, compensation No., shape compensation amount, tool length wear amount</p>
(3) Cycle time display (1-part system display only)	<p>This displays the automatic operation time and cycle time.</p> <p>The displayed items are switched with integrated time window.</p>
(4) Completed workpiece display (1-part system display only)	<p>This displays the number of workpieces which have been completed.</p> <p>The display follows the machining parameters "#8001 WRK COUNT M" to "#8003 WRK COUNT LIMIT".</p>
(5) Machine status animated display (1-part system display only)	<p>This displays the current tool No., tool type, next command movement direction, coordinate rotation status, mirror image status, and spindle rotation direction/coolant status.</p> <p>Note that the spindle rotation direction/coolant status display differ according to the machine tool builder.</p>
(6) M, S, T, B commands	<p>This displays the values command for M (miscellaneous function command value), T (tool command value), B (2nd miscellaneous function command value) and S (spindle command rotation speed).</p> <p><1-part system display> The displayed number of M (miscellaneous function command value) depends on parameter "#12005 Mfig". The displayed number of T (tool command value) and B (2nd miscellaneous function command value) is "1" regardless of the parameters "#12009 Tfig" and "#12011 Bfig". However, the display/non-display of B (2nd miscellaneous function command value) depends on parameter "#1170 M2name". Refer to "Manual numerical value command" for the manual numerical value commands.</p> <p>This displays the S command and spindle actual rotation speed value about S (spindle command rotation speed). S command is always displayed. However the spindle rotation speed might not be displayed by screen area constraints.</p> <p><2-part system simultaneous display> The active area MSTB display can be changed by the menu MST Chg. (Refer to "2.1.3 Operation of 2-part System Simultaneous Display" for detail.)</p>
(7) Speed display	<p>In interpolation feed: This displays the vector direction speed currently being moved in.</p> <p>In each axis independent feed: This displays the speed of the axis with highest speed. The actual machine feedrate is displayed with the parameters "#1125 real_f".</p>

Display item	Details
(8) Spindle/Wait display (1-part system display only)	The spindle tool No. and the standby tool No. are displayed. These displayed contents differ according to the machine tool builder.
(9) Load meter display (1-part system display only)	The spindle load and Z axis load, etc., are displayed in a bar graph. If Spindle/standby display is not being used, a load meter can be displayed in the Spindle/standby display area. These displayed contents differ according to the machine tool builder.
(10) Machining program currently being executed (Note)	
Main 010...	This displays the program No., sequence No. and block No. currently being executed.
Sub 01234...	When a subprogram is being executed, this displays the subprogram's program No., sequence No. and block No.
(11) Buffer display	This displays the contents of the machining program currently being executed. The block being executed is highlighted.

(Note) If the program No. (program name) exceeds 12 characters, "*" will appear at the 12th character.

Menus

Menu	Details	Reference
 Search	This executes operation search.	2.2 Operation Search
 Reserch	This executes restart search.	2.3 Restart Search
 Edit	This edits the machining program in search.	2.4 Program Edit
 Trace	This executes trace.	2.5 Trace
 Check	This checks the program. This menu does not appear if the program check function option is not provided.	2.6 Program Check (2D) 2.7 Program Check (3D)
 Cnt exp	This enlarges the counter, and displays the counters for all axes.	2.8 Counter All-axis Display
 Offset	This displays the tool compensation amount. A value can also be set for the tool compensation amount.	2.9 Tool Offset Amount
 Coord	This displays the workpiece coordinate system offset. A value can also be set for the workpiece coordinate system offset.	2.10 Workpiece Coordinate System Offset
 Cnt set	A arbitrary value can be set in the relative position counter.	2.11 Counter Set
 MST	This executes a manual numerical value command.	2.13 Manual Numerical Value Command
 Modal	This displays the program modal.	2.14 Modal Display
 Tree	This displays the program tree.	2.15 Program Tree Display

Menu	Details	Reference
 Time	This displays the date, time and integrated time, etc., The date, time and integrated time, etc., can also be set.	2.16 Integrated Time Display
 Com var	This displays the common variables. A value can also be set for the common variable.	2.17 Common Variables
 Loc var	This displays the local variables.	2.18 Local Variables
 P corr	This corrects the buffer.	2.19 Buffer Correction
 PLC SW	This turns the PLC switches ON or OFF.	2.20 PLC Switch Function
 G92 set	This sets and cancels the origin.	2.12 Origin Set, Origin Cancel
 Col stop	This executes verify stop.	2.21 Verify Stop
 LdMeter	The spindle load and Z axis load, etc., are displayed in a bar graph. These displayed contents differ according to the machine tool builder.	2.22 Load Meter Display
 Sp-stby	The current spindle tool No. and the standby tool No. are displayed. These displayed contents differ according to the machine tool builder.	2.23 Spindle, Standby Display
 TipDisp	This displays the tool center point coordinates, handle interruption amount (tool axis movement), and tool center point speed. (Note) If the 5-axis related options (tool center point control, tool length compensation along the tool axis, tool handle feed & interrupt) are not mounted, these menus are not displayed.	2.24 Tool Center Point Display
 All Sp	This displays commanded/actual rot speed values of all the spindles.	2.25 All Spindles' Rotation Speed Display
 MST Chg	This switches MSTB display for the active area. (Note) This menu is displayed only when number of the part system are 2 or more and the 2-part system simultaneous display ("#11019 2-system display" = "1") is enabled.	2.1.3 Operation of 2-part System Simultaneous Display

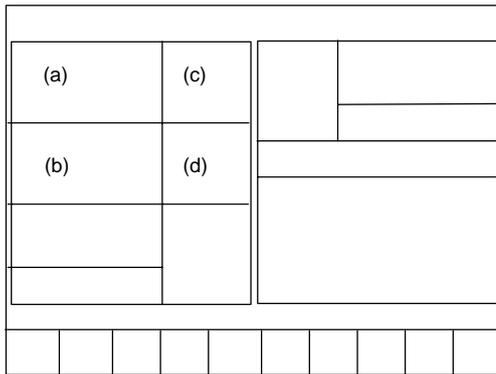
2.1.1 Counter Display

The counter display contents of 1-part system display and 2-part system simultaneous display are as follows.

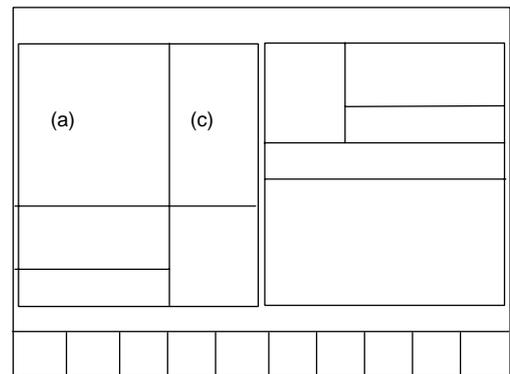
For 1-part system display

(1) Auto/MDI

<When the counter displayed axes are five or less>

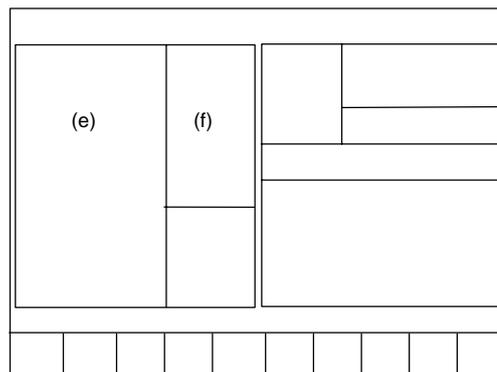


<When the counter displayed axes are six or more>



Layout	5-axis related specifications disabled	5-axis related specifications enabled	Remarks
(a)	Relative position	Tip workpiece position	This can be changed with "#8901 counter type 1".
(b)	Program position	Program position	This can be changed with "#8902 counter type 2".
(c)	Remaining command	Remaining command	This can be changed with "#8903 counter type 3".
(d)	Next command	Next command	This can be changed with "#8904 counter type 4".

(2) Manual

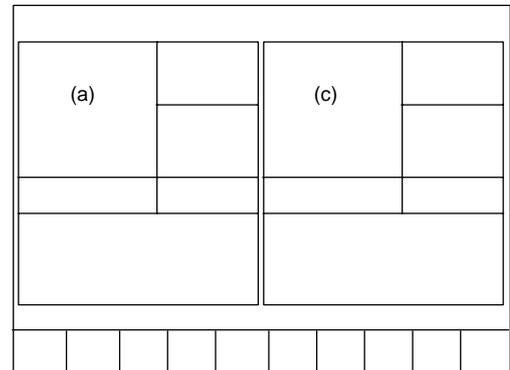
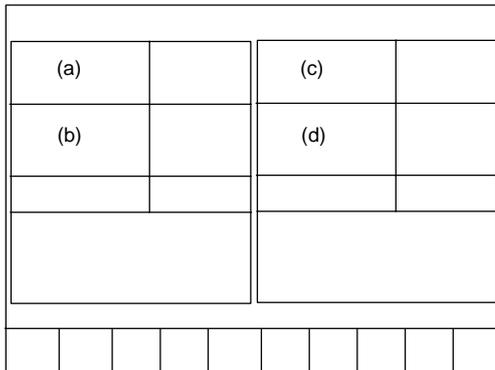


Layout	5-axis related specifications disabled	5-axis related specifications enabled	Remarks
(e)	Relative position	Tip workpiece position	This can be changed with "#8905 counter type 5".
(f)	Machine position	Machine position	This can be changed with "#8906 counter type 6".

For 2-part system simultaneous display

<When the counter displayed axes are four or less>

<When the counter displayed axes are five or more>



Layout	5-axis related specifications disabled	5-axis related specifications enabled	Remarks
(a)	Relative position	Tip workpiece position	This can be changed with "#8901 counter type 1".
(b)	Program position	Program position	This can be changed with "#8902 counter type 2".
(c)	Relative position	Tip workpiece position	This can be changed with "#8903 counter type 3".
(d)	Program position	Program position	This can be changed with "#8904 counter type 4".

2.1.2 Changing Between <Auto/MDI> and <Manual> [1-part System Display Only]

The correspondence between the mode selection switch settings and display contents on the left side of the screen is as follows.

Mode selection switch setting	Screen display
No mode	Auto/MDI
Memory	
Tape	
MDI	
Jog	Manual
Handle	
Step	
Manual arbitrary feed	
Reference position return	
Automatic dog-less reference position return	
Rapid traverse	
Jog + handle	
Rapid traverse + handle mode	

2.1.3 Operation of 2-part System Simultaneous Display

Changing active area

The active area can be changed by tab key ( ) or touching the area on touch panel.

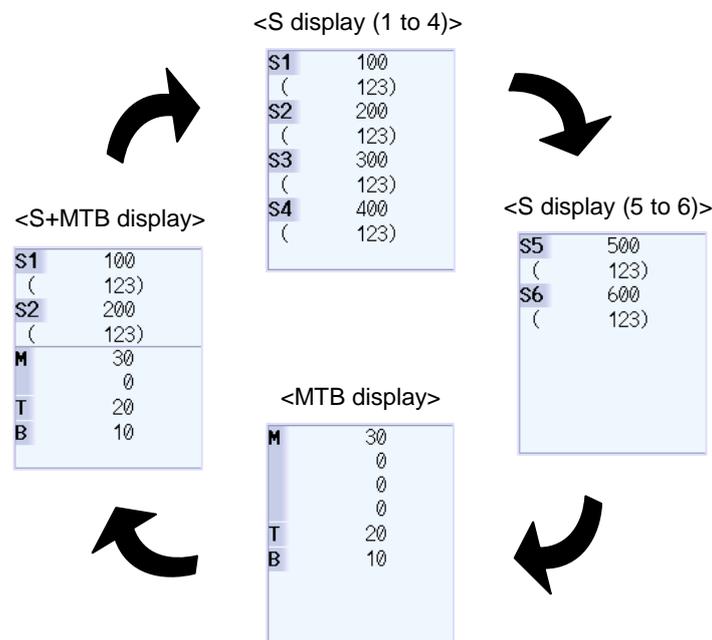
(Note 1) The active area cannot be changed while displaying a pop-up window.

(Note 2) The active area can be changed while displaying the S/W keyboard.

Changing MSTB display

The MSTB display can be changed to the active area by the main **MST Chg** menu on the monitor screen of 2-part system simultaneous display.

The MSTB display changes as follows every time the main menu is pressed.



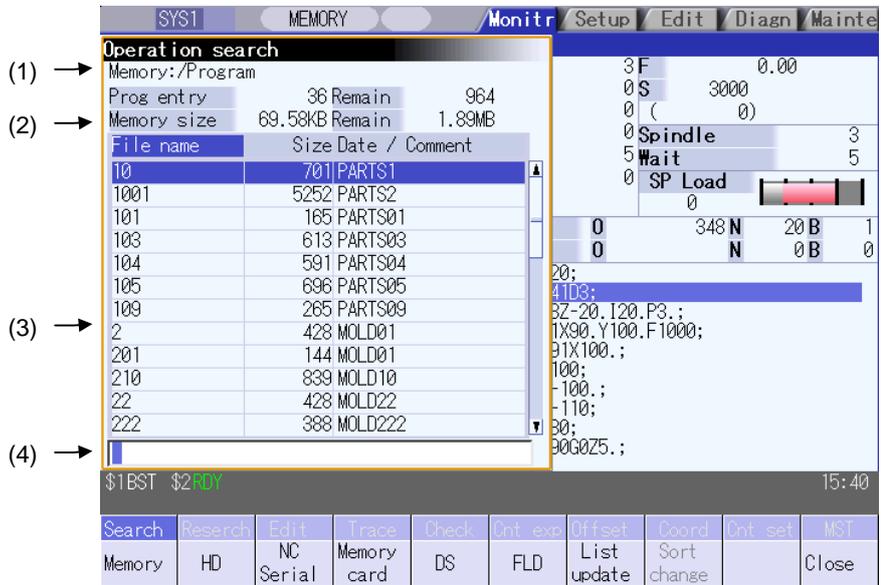
The above S (spindle command rotation speed) display of <MSTB display> is designated by the parameter "#8925 SP on 1st part sys" to "#8928 SP on 4th part sys". The number of spindle which can be specified in each part system is "2". The maximum display number of M (miscellaneous function command value) is "2" regardless of the parameter "#12005 Mfig".

The maximum display number of T (tool command value) or B (2nd miscellaneous function command value) is "1" regardless of the parameter "#12009 Tfig" or "#12011 Bfig". However, the display/non-display of B (2nd miscellaneous function command value) depends on the setting of the parameter "#1170 M2name".

2.2 Operation Search



On this screen, the program can be called from the program storage site, such as a memory, by designating the program (program No.) to be automatically run and the program start position (sequence No., block No.). In 70 series, only the memory, the serial, and the memory card can be used for the input/output device. When the multi-part system program management is valid (#1285 ext21/bit0 =1, ext21/bit1 =1), for only the machining program in memory card, the program search is executed across all part systems in batch.



Display items

Display item	Details
(1) Device name, directory display	This displays the device name and directory designated when the program was selected.
(2) Capacity display	This displays the capacity of the device displayed in (1).
(3) List of directories and files	This displays a list of the contents contained in the device or directory displayed in (1). Use and to scroll the displayed list. If the device is an HD, FD, memory card or DS, the file update date/time is displayed in the <Date/Comment> field. For the NC memory, the machining program comment is displayed. Whether to show or hide the comment field can be selected by pressing the menu. When the comment field is hidden, the file name field will be enlarged. (Up to 13 characters can be displayed in the file name field when the comment is shown, and 32 when the comment is hidden.) If the file name exceeds the maximum number of characters, "*" will appear at the last character.
(4) Input area	This displays the input key details.

Menus

Menu	Details	Type	Reference
Memory	This selects the device for searching for the program. When a device with directory is selected, the route is selected first. (In 70 series, only "Memory", "Serial", and "Memory card" are displayed.)	C	2.2.1 Executing an Operation Search
HD		C	
Serial		C	
Memory card		C	
DS		C	
FD		C	
List update	This updates the list contents. (The latest contents of the currently selected device and directory are listed.)	C	
Sort change	This changes the method that the list is sorted.	C	2.2.3 Changing the Sorting Method
Comment nondisp	This changes whether to show or hide the comment field in the list. When the comment field is hidden, the file name field will be enlarged.	B	2.2.2 Changing Whether to Show or Hide the Comment Field
Close	This closes the pop-up window and quits this function.	C	

(Note 1) Some items may not be displayed depending on the device.

○: Displayed ×: Not displayed.

Device	Memory	HD	Serial	Memory card	DS	FD
Prog entry	○	○	×	○	○	○
Remain	○	×	×	×	×	×
Memory side	○	○	×	○	○	○
Remain	○	○	×	○	○	○
List	○	○	×	○	○	○

(Note 2) When using serial, the port number set with parameter "#9005 TAPE MODE PORT" is connected, and search.

2.2.1 Executing an Operation Search

Operation method

- (1) Select the part system to run with the keys. → The selected part system appears on the upper left of the screen.
- (2) Press the main menu . → The sub-menu appears.
The list appears as a pop-up window.
- (3) Select the device.
(Example) → The selected device name and route directory (HD:/) appear in the Device name, Directory display field.
- (4) Using the , , , and keys, move the cursor to the directory containing the file to be set. → The following is displayed.

```

          DIR
.         DIR
..        DIR
ABC       DIR
SRAM.BIN 1019904 Mar 06 14:54 2003

```

If the list contents differ from the actual device or directory, press the menu .

- (5) Press the key. → The active window changes to the directory.
- (6) Using the , , , and keys, move the cursor to the target machining program.

```

          DIR
          DIR
123.PRG   62 Dec 20 15:24 2002
68.PRG    62 Dec 20 15:24 2002
69.PRG    166 Dec 20 15:24 2002

```

- (7) Press the key. → The search starts.
When the search is completed, the message "Search Complete" appears.
The searched device and program position appear in the field for displaying the machining program currently being executed.
The list closes, and the main menu appears again.

When designating the sequence number and block number and searching for the program, input the search conditions in the input area. The program No. is categorized according to the presence of an extension.

Input details	Program No.	Sequence No.	Block No.
1001/1/2	O1001	N1	B2
1001.PRG/1/2	O1001.PRG	N1	B2
/1/2	(Currently searched O No.)	N1	B2
1001//2	O1001	N0	B2
1001/1	O1001	N1	B0
1001	O1001	N0	B0
/1	(Currently searched O No.)	N1	B0
//2	(Currently searched O No.)	N0	B2

(Note 1) Program after operation search

- (a) Operation search is cancelled if the following operations are carried out on the Edit screen after operation search. Execute restart search in this case.
 - If the searched program is erased.
 - If the searched sequence No. is erased.
 - If the block corresponding to the searched block No. is erased.
- (b) Operation search cannot be executed in another block during MDI operation even if the block is stopped.
Wait for MDI operation to end or reset the NC before searching.

(Note2) A program with 33 or more file name characters cannot be searched.

2.2.2 Changing Whether to Show or Hide the Comment Field

The file name field can be extended by changing whether to show or hide the comment field.

Operation method

- (1) Press the menu "Comment nondisp".
- ➔ The file name is displayed up to 32 alphanumeric characters and symbols after the comment field disappears. The menu is highlighted.
- (2) Press the highlighted "Comment nondisp" again.
- ➔ The comment field appears, and the file name is displayed up to 15 alphanumeric characters and symbols. The menu highlight returns to normal.

(Example) When the file name is "123456789A123456789B123456789C12"

"Comment display" is validated: The file name display is omitted and "*" is attached.

File name	Size	Date / Comment
123456789A12*	126	COLOR_CHK_01

"Comment display" is invalidated: The file name is displayed up to 32 alphanumeric characters and symbols.

File name	Size
123456789A123456789B123456789C12	126

(Note 1) The set status is held even if the NC power is turned OFF.

(Note 2) The set status is common for list view of operation screen, restart search screen, etc.

(Note 3) The cursor moves to the head when display/non-display in the comment field is switched.

2.2.3 Changing the Sorting Method

Sorting method changes to 1→2→3→4→5→1 •••• every time the Sort change menu is pressed.

The selected sorting method is also displayed in the list on the other screens. The method is common for all the devices, as well. The sorting method will be held even after the power is turned OFF and ON.

Up to 64 files (Including current directory "." and one directory above "..") can be sorted within a directory. If exceeds 65 files, the Sort change menu is displayed in gray and changing the sorting method is impossible. In this case, NC memory program is displayed with "1. File name No. ascending order". Other devices are displayed with "2. File name character code ascending order".

No.	Sorting method	Priority
1	File name No. ascending order	1. "." for current directory, ".." for one directory above 2. Numerical value (excluding the case where "0" is put at the beginning) ascending order 3. Character code ascending order
2	File name character code ascending order	1. "." for current directory, ".." for one directory above 2. Character code ascending order
3	File name character code descending order	1. Character code descending order 2. "." for current directory, ".." for one directory above
4	Comment ascending order	"Date/comment" character code ascending order
5	Comment descending order	"Date/comment" character code descending order

(Note 1) The character code ascending (descending) order is the method in which file names are compared one by one using the ASCII code.

Example: If the ascending order is applied, the appropriate order would be 1→A as "1" equals to "0x31" and "A" equals to "0x41".

(Note 2) When the device is "Memory", display for "current directory" and "one directory above" is not available.

(Note 3) When comments are the same, sorting is carried out by the file name character code ascending (descending) order.

(Note 4) "Date/comment" for the devices other than "Memory" are compared using character string only. Thus, the order will not be one by the actual date.



2.3 Restart Search

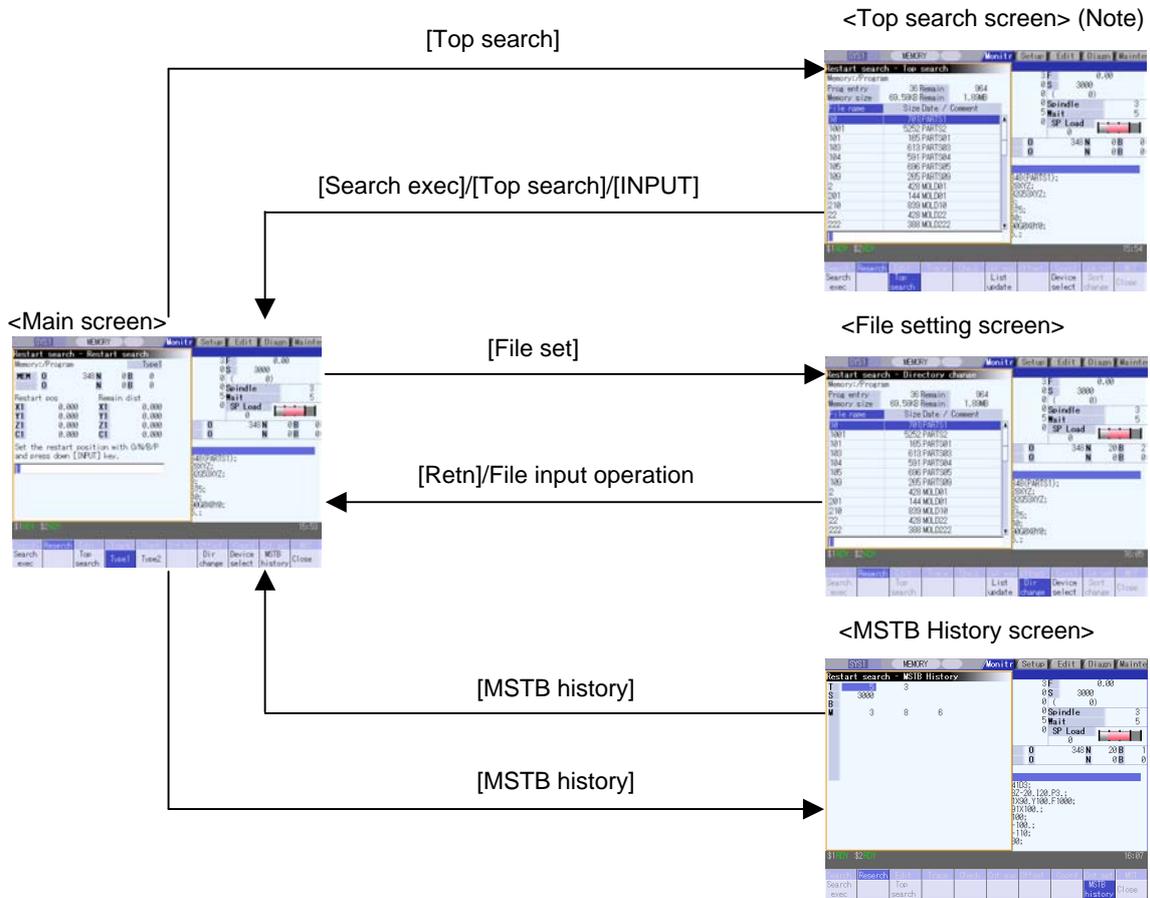
If machining is temporarily stopped due to tool breakage, etc., the program restart function searches for the block of the machining program to be restarted, and restarts machining from that block.

There are two types of restart, type 1 and type 2.

Restart method	Details	
Restart type 1	After machining is reset due to a tool breakage, etc., machining is restarted from the designated sequence number and/or block number. Only the program which had been executed just before can be restarted. Even after the power is turned ON again, the program can be restarted if the program has been executed before the power supply is turned OFF. (Note 1) In case that NC has been reset or the power supply has been turned OFF while fixed cycle, custom fixed cycle, machine tool builder macro, or macro interruption is executed, the search with no input to the setting will be performed for each source program. (Note 2) When the user macro is executed, the user macro execution position is searched.	
Restart type 2	Automatic top search OFF	A top search must be executed from the screen. Then, command a sequence No. and block No., and restart a program.
	Automatic top search ON	A top search may not be executed from the screen. A top search is automatically executed, so it is not necessary to execute a top search from the screen. Command a sequence No. and block No., and restart a machining. An arbitrary program can be restarted by designating the program No. When a program No. is omitted, the program currently searched is restarted.

The validity of the automatic top search can be changed with control parameter "#8914 Auto Top search".

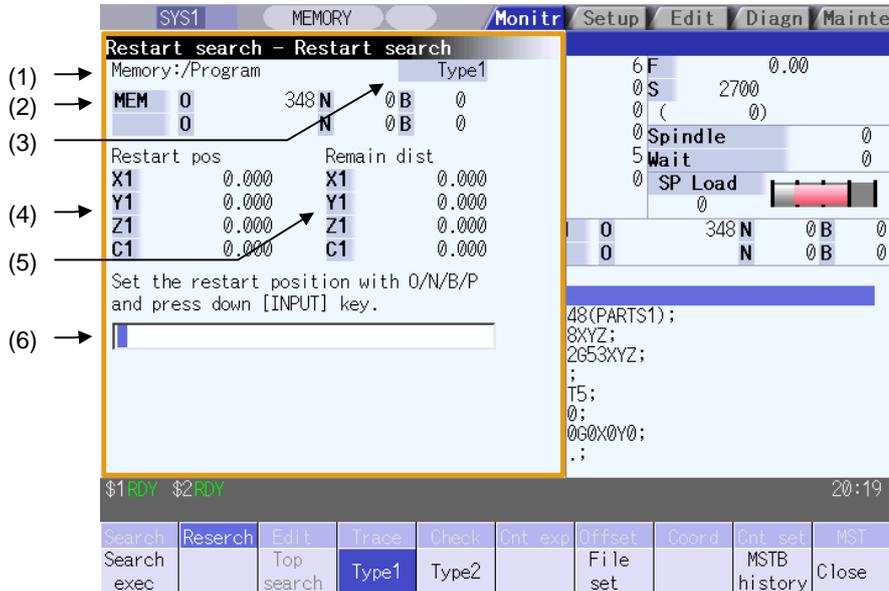
Screen transition



- (Note)** In the following cases, the screen cannot be switched to top search screen.
- Restart type 1
 - The parameter "#8914 Auto Top search" is "1" (automatic top operation)

2.3.1 Main Screen

The type 1 and type 2 restart search can be executed from the Main screen.



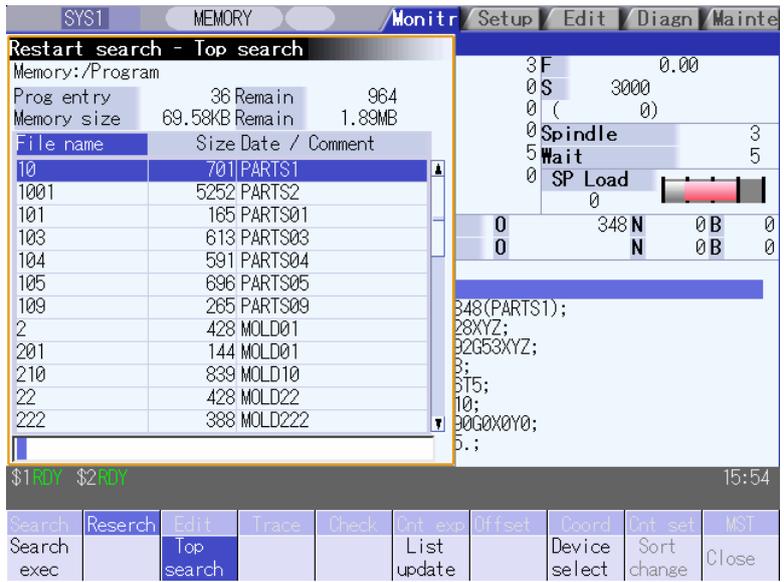
Display items

Display item	Details
(1) Device and directory display	This displays the device and directory where the searched machining program is located.
(2) Research position	This displays the researched main program position (program No., sequence No., block No.).
(3) Restart type	This displays the restart search type.
(4) Position when restart search is completed	This displays the position on the local coordinate system when the restart search is completed.
(5) Remaining distance when restart search is completed	This displays the remaining distance when the restart search is completed.
(6) Input area	This displays the input key details.

Menus

Menu	Details	Type	Reference
Search exec	This starts the restart search based on the designated device, directory, program number (O), sequence number (N), block number (B) and number of block execution times (P).	C	
Top search	This changes to the pop-up window for executing top search, and enables top search. When the type 1 is selected or the parameter "#8914 Auto Top search" is set, this menu cannot be selected.	B	2.3.2 Top Search Screen
Type1 Type2	This selects the restart search type. Restart search is executed with the highlighted restart type. The type 1 or type 2 menu is always highlighted. When restart search is executed, the selected restart type is displayed at the display item "(3) Restart type".	B	
File set	This changes to the pop-up window for setting the file, and enables the file to be selected.	B	2.3.3 File Setting Screen
MSTB history	This opens the MSTB history screen as a pop-up window. The M, S, T and B command used in the machining program are listed on the MSTB history screen. If the cursor is moved to the listed M, S, T or B command and the <input type="text" value="INPUT"/> key is pressed, that command will be executed.	B	2.3.4 MSTB History Screen
Close	This closes the pop-up window and quits this function.	C	

2.3.2 Top Search Screen



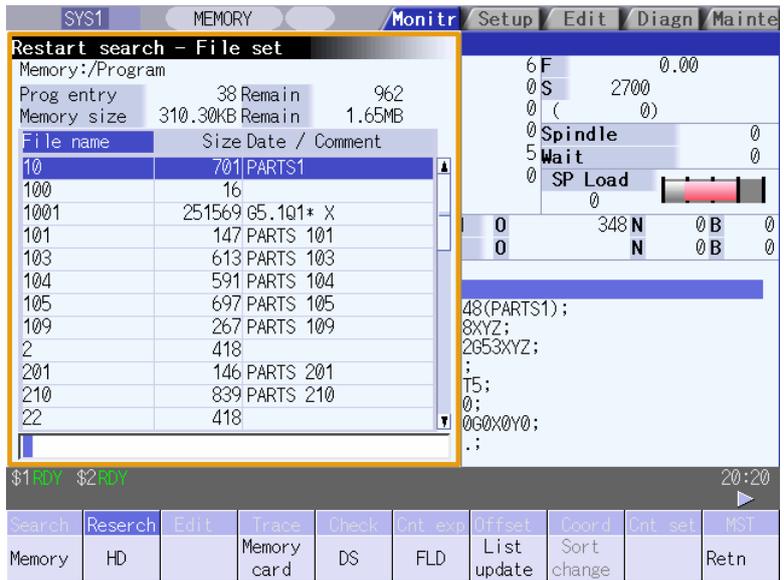
Display items

Same as "2.2 Operation Search".

Menus

Menu	Details	Type	Reference
Search exec	This starts the restart search based on the designated device, directory, program number (O), sequence number (N), block number (B) and number of block execution times (P).	C	
Top search	When the menu is highlighted, it indicates that the top search pop-up window is open. When this is highlighted, it indicates that the top search pop-up window is open. If pressed again when highlighted, the top search pop-up window closes, and the main pop-up window opens.	B	
List update	This updates the list details. (A list of the latest details of the currently selected device and directory is displayed.)	C	
Device select	This displays the device menu. Select which device program to search for from this menu. If a device having a directory is selected, the root is selected first.	C	
Sort change	This updates the list sorting method.	C	2.2.3 Changing the Sorting Method
Comment nondisp	This changes whether to show or hide the comment field in the list. When the comment field is hidden, the file name field will be enlarged.	C	2.2.2 Changing Whether to Show or Hide the Comment Field

2.3.3 File Setting Screen



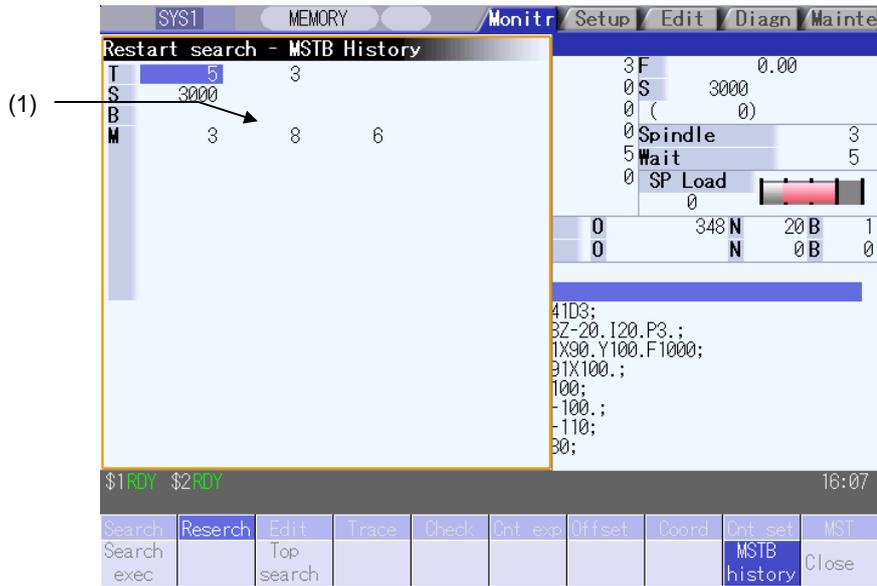
Display items

Same as "2.2 Operation Search".

Menus

Menu	Details	Type	Reference
Memory	Select the device to search program. When a device other than memory is selected, the root is selected first. (In 70 series, only "Memory" and "Memory card" are displayed.)	C	
HD		C	
M-Card		C	
DS		C	
FD		C	
List update	This updates the list details. (A list of the latest details of the currently selected device and directory is displayed.)	C	2.2.3 Changing the Sorting Method
Sort change	This updates the list sorting method.	C	
Retn	This returns the main menu.	C	
Comment nondisp	This changes whether to show or hide the comment field in the list. When the comment field is hidden, the file name field will be enlarged.	B	

2.3.4 MSTB History Screen



Display items

Display items	Details
(1) MSTB history	The M, S, T and B command used in the machining program are listed. Up to 35 M commands, 3 commands each for S1 to S6, 3 T commands and 3 B commands are displayed.

Menus

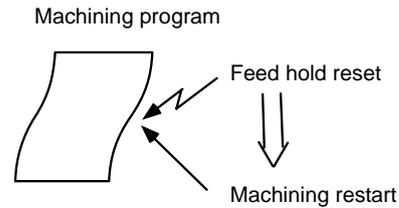
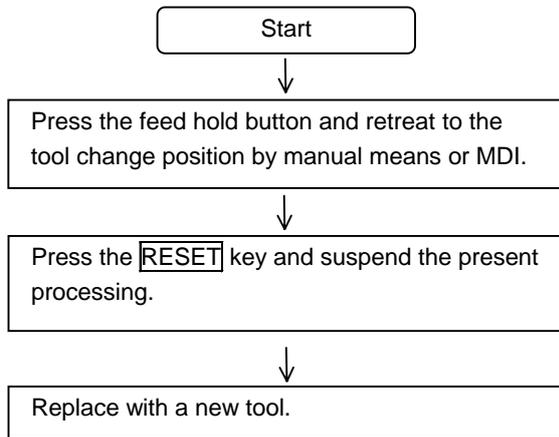
Menu	Details	Type	Reference
MSTB history	When this is highlighted, it indicates that the MSTB history pop-up window is open. If the menu is pressed when this menu is highlighted, the system will close the MSTB history pop-up window, and the main pop-up window opens.	B	

2.3.5 Operation Sequence for Program Restart

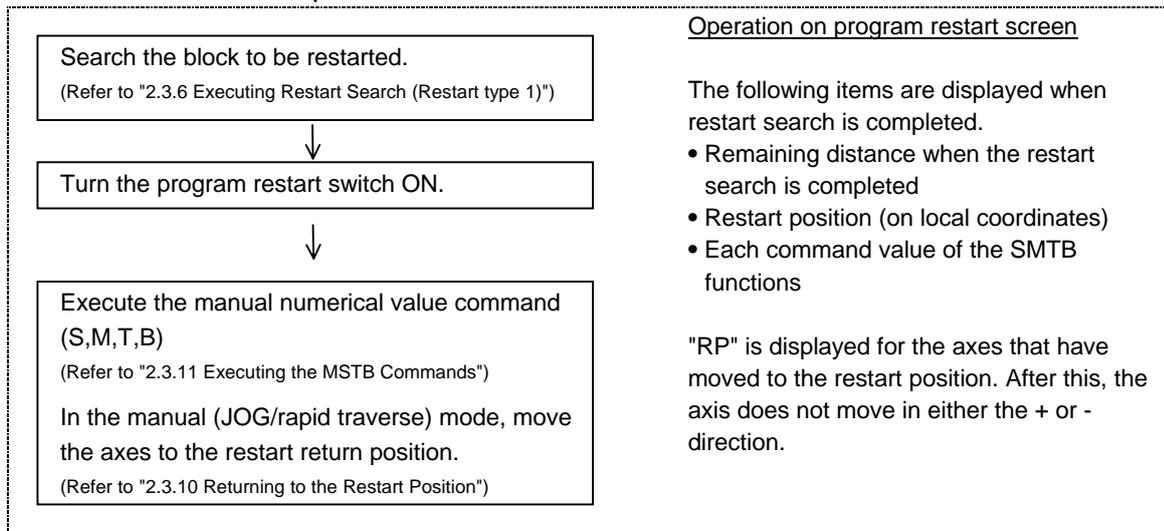
There are two types of restart, type 1 and type 2.

Restart type 1

When feed hold has been applied and reset because the tool has broken, etc., restart with Restart type 1.



If the tool compensation amount (tool length, tool radius) is changed, change the tool on tool compensation screen.

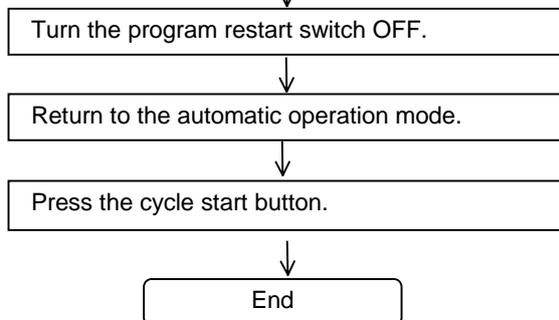


Operation on program restart screen

The following items are displayed when restart search is completed.

- Remaining distance when the restart search is completed
- Restart position (on local coordinates)
- Each command value of the SMTB functions

"RP" is displayed for the axes that have moved to the restart position. After this, the axis does not move in either the + or - direction.



If the cycle starts when the "Automatic return by program restart" parameter ("#1302 AutoRP") is ON, the processing is restarted after all axes return to the restart position at the same time.
(Refer to "2.3.10 Returning to the Restart Position")

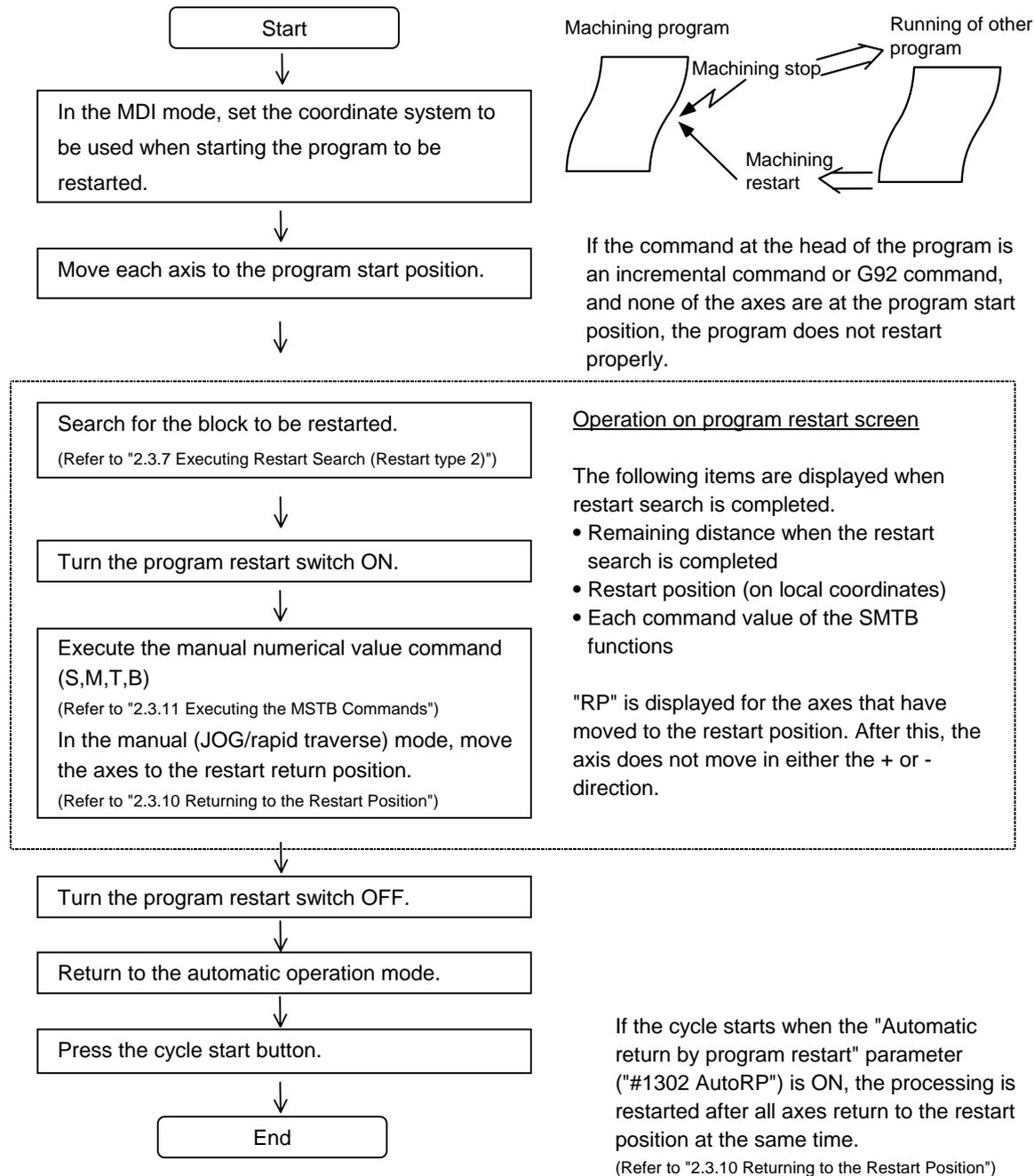
- (Note 1)** When single block signal is OFF, the machining is restarted without stopping temporarily after the axes return to the restart position. (The cycle starts when the automatic return by program restart is valid.)
- (Note 2)** Execute Reset & Rewind for resetting.
- (Note 3)** Restart & Search must be carried out with Restart Type 2 when the power has been turned ON again.
If Restart Type 1 is selected, "Search error" occurs.
- (Note 4)** Restart Search is not possible for a program with 33 or more file name characters.

Restart type 2

If a machining program differing from the machining program to be restarted was run before starting restart search, restart with restart type 2.

When the coordinate system of the automatic operation last time and the coordinate system of the machining restart are changed, it is possible to restart.

(Note) The necessary matters for starting the machining program, such as setting the coordinate system, must be completed before starting restart search.



(Note 1) When single block signal is OFF, the machining is restarted without stopping temporarily after the axes return to the restart position. (The cycle starts when the automatic return by program restart is valid.)

(Note 2) Restart Search is not possible for a program with 33 or more file name characters.

(Note 3) When the searched program do not exist, the operation message "Search error" appears.

2.3.6 Executing Restart Search (Restart Type 1)

When feed hold has been applied and reset because the tool has broken, etc., restart with Restart type 1. When the multi-part system program management is valid (#1285 ext21/bit0 =1, ext21/bit1 =1), the program search is executed across all part systems in batch.

However, only when the following requirements are met, this is valid.

- It is the machining program in memory card.
- ONBP is not used.

Operation method

(Example) When tool breakage during execution of O1000 N7 occurred, and restarting from the O1000 N6 block

- (1) Press the feed hold button and retreat to the tool change position by manual means or MDI.
Press the reset key and suspend the present processing.
- (2) Replace with a new tool.
- (3) When using tape operation, index the top of the tape.
- (4) Press the main menu . → The submenu appears.
The main screen for restart search appears as a pop-up window.
- (5) Set the position to restart search in the setting area. → Delimit the ONB number in the setting area using /.
<When O No. was attached>
The main or sub program is targeted.
(Ex.) 1000/6/0
<When O No. was not attached>
The program currently searched is targeted.
(Ex.) /6/0
(Note) When the key is pressed without entering data in the input area, restart search will be carried out for the last execution block.
(Type 1 only)
- (6) Press the or menu. → Restart search is executed.
The message "In restart search" appears during the search process, and the message "Restart search complete" appears when completed.
Each axis' restart position and the restart remaining distance are displayed.
When the menu is pressed, the MSTB history screen will open as a pop-up window, and the M, S, T, B command used in the machining program will be listed.

2.3.7 Executing Restart Search (Restart Type 2)

If a machining program differing from the machining program to be restarted was run with tape, memory or HD operation before starting restart search, restart the respective machining program with restart type 2. The restart type 2 operation sequence is the same as restart type 1, but necessary matters for starting the machining program, such as setting the coordinate system, must be completed before starting restart search. When the parameter "#8914 Auto Top search" is "0", execute the top search for the machining program.

Operation method (When the parameter "#8914 Auto Top search" is "0")

(Example) When restarting from subprogram O123 N6 B2 called from main program O1000

- (1) Turn the power ON, and return all axes to the reference position.
- (2) In the MDI mode, set the coordinate system to be used when starting the program to be restarted.
- (3) Move each axis to the program restart position.
- (4) When using tape operation, index the top of the tape.
- (5) Press the main menu . ➔ The submenu appears.
The main screen for restart search appears as a pop-up window.
- (6) Press the .
- (7) Press the sub-menu . ➔ The window for top search appears as a pop-up window.
- (8) Using the , , , and keys, move the cursor to the target machining program. ➔ Set the cursor to 1000.
- (9) Press the key. ➔ The top search starts.
When the top search is completed, the message "Search Complete" appears.
The top search pop-up window closes, and the main screen for restart search appears as a pop-up window.
The search can be executed by pressing the menu instead of the key.

- (10) → Delimit the ONBP number in the setting area using /.
 <When O No. was attached>
 The restart search is executed at designated position by NBP No. with the set O No. at the head.
 (Ex.) 123.PRG/6/2/1
 <When O No. was not attached>
 The program currently searched is targeted.
 (Ex.) /6/2/1

P sets the number of times that the block targeted for the restart search appears.

For example, if a block in a subprogram is searched, and the subprogram is called out several times, the block to be searched is also executed several times. Thus, which execution block to be searched must be set. (If "0" is set, it is handled in the same manner as "1".)

To search the first execution block, or to search for a block executed only once, this item does not need to be set.

- (11) → Restart search is executed.
 The message "In restart search" appears during the search process, and the message "Restart search complete" appears when completed.
 Each axis' restart position and the restart remaining distance are displayed.
 When the menu is pressed, the MSTB history screen will open as a pop-up window, and the M, S, T, B command used in the machining program will be listed.

Operation method (When the parameter "#8914 Auto Top search" is "1")

(Example) When restarting from subprogram O123 N6 B2 called from main program O1000

- (1) Turn the power ON, and return all axes to the reference position.
- (2) In the MDI mode, set the coordinate system to be used when starting the program to be restarted.
- (3) Move each axis to the program restart position.
- (4) When using tape operation, index the top of the tape.
- (5) Press the main menu **Restart search**. → The submenu appears.
The main screen for restart search appears as a pop-up window.
- (6) Press the **Type 2**. →
- (7) Press the sub-menu **File set**. → The window for file set appears as a pop-up window.
- (8) Using the **↑**, **↓**, **↶** and **↷** keys, move the cursor to the target machining program. → Set the cursor to 1000.
- (9) Press the **INPUT** key. → The screen for restart search appears as a pop-up window.

- (10) → Delimit the ONBP number in the setting area using /.
<When O No. was attached>
The restart search is executed at designated position by NBP No. with the set O No. at the head.
(Ex.) 123.PRG/6/2/1
<When O No. was not attached>
The program currently searched is targeted.
(Ex.) /6/2/1
- (11) → Restart search is executed.
The message "In restart search" appears during the search process, and the message "Restart search complete" appears when completed.
Each axis' restart position and the restart remaining distance are displayed.
When the menu is pressed, the MSTB history screen will open as a pop-up window, and the M, S, T, B command used in the machining program will be listed.

2.3.8 Changing the Device

Operation method

- (1) Press the submenu . → The window for file setting appears as a pop-up window.
- (2) Select the device, and press the menu key.
(Example) → The selected device name and root directory (HD:/) appear in the Device name and Directory display field.

(Note) In 70 series, only the memory and the memory card can be used for the input/output device.

2.3.9 Changing the Directory with the Main Screen

Operation method

- (1) Press the submenu . → The window for file setting appears as a pop-up window.
- (2) Using the , , , and keys, move the cursor to the directory to be moved. → The following is displayed.

.	DIR
..	DIR
ABC	DIR
SRAM.BIN	1019904 Mar 06 14:54 2003

If the list contents differ from the actual device or directory, press the menu .

- (3) Press the key. → The directory changes, and the contents of the designated directory appear in the list.
The moved directory appears in the directory display field.

.	DIR
..	DIR
123.PRG	62 Dec 20 15:24 2002
68.PRG	62 Dec 20 15:24 2002
69.PRG	166 Dec 20 15:24 2002

- (4) Repeat steps (2) and (3).
- (5) After changing the directory, press the or key. → The restart search main window opens as a pop-up window.

2.3.10 Returning to the Restart Position

The axis is returned to the restart position after restart search is completed.

The method for returning to the restart position (manual/automatic) can be selected with the parameters "#1302 AutoRP".

- 0 : Manual restart position return
- 1 : Automatic restart position return

Operation method (manual restart position return)

- (1) Turn the restart switch ON.
- (2) Enter the manual (JOG/rapid traverse) mode.
- (3) Move the axis in the restart return direction.



The restart position and "RP" appear sequentially from the axes that have been returned. The Restart remaining distance is "0".

Restart pos		Remain dist	
X1	40.000	X1	40.000
Y1	150.100	Y1	150.100
Z1	-70.000	Z1	-25.000
C1	0.000RP	C1	0.000

- (4) When all axes have been returned, turn the restart switch OFF.

- (Note 1)** When the restart switch is ON, move the axis in the same direction as the restart direction. If moved in the reverse direction, the operation error "M01 R-pnt direction illegal" occurs. If the tool needs to be retracted once, such as if the tool is interfering with the workpiece, turn the restart switch OFF and retract the axis manually.
- (Note 2)** After restart position return is completed, the axis cannot be moved if the restart switch is ON. If the axis is moved, the operation error "M01 restart switch ON" occurs.
- (Note 3)** If even one axis has not completed return to the restart position at cycle start, the error "T01 Restart pos. return incomplete" occurs. Note that if the axis has been returned to the restart position once and is not at the restart position during cycle start, the alarm does not occur.
- (Note 4)** If the axis to be returned to the restart position is a machine lock axis, the operation error "M01 program restart machine lock" occurs. Release the machine lock before returning to the restart position.
- (Note 5)** If the restart switch is turned to ON after the axis is returned to the restart position with the restart switch OFF, "PR" may not be displayed. Return to the restart position after the restart switch is turned ON.
- (Note 6)** After restart search had been completed, if the movement command is issued by MDI before automatic restart position return has been started, the program error (P48) occurs. Perform the operation again after reset.

- (Note 7)** After restart search had been completed, if the T command different from the program is issued by MDI before the searched program has been started, the program error (P48) may occur. Perform the operation again after reset. T commands which result in errors are shown below. (Only L system)
- (1) When "#1100 Tmove" = "0"
T command that selects a tool with different tool length/wear amount than the tool selected by the program
 - (2) When "#1100 Tmove" = "1"
A program error will not occur by T command.
 - (3) When "#1100 Tmove" = "2"
T command that selects a tool with different wear amount than the tool selected by the program
- (Note 8)** After restart search had been completed, if the handle interruption is performed before the searched program has been started, always the same operation as manual absolute ON is executed regardless of "Manual absolute" signal.

Automatic restart position return

If the parameter "#1302 AutoRP" is set to "1", each axis returns to the restart position with dry run at cycle start. Machining restarts after returning. The order that the axes return follows parameter "#2082 a_rstax".

- (Note 1)** Manually move the axis to a position where the tool does not interfere with the workpiece before starting the cycle.
- (Note 2)** Even if the parameter "#1302 AutoRP" is set to 1, the axes can be returned manually to the restart position by turning the restart switch ON. In this case, move the axes in the order of manual restart position return → automatic restart position return. After completing automatic restart position return, if the operation is stopped temporarily and the restart switch is turned ON, the operation error "M01 restart switch ON" occurs.
- (Note 3)** Once the axis has been manually returned to the restart position, if it is moved from the restart position, it will not return to the restart position even if automatic restart position return is executed.
- (Note 4)** The axis for which parameter "#2082 a_rstax" is set to 0 does not return to the restart position. Note that if 0 is set for all axes, all axes simultaneously return to the restart position. Designate the axes in part system units. If the axis for which parameter "#2082 a_rstax" is set to 0 has not completed manual restart position return when automatic restart position return is started, the error "T01 Restart pos. return incomplete" occurs.
- (Note 5)** After automatic restart position return had been started, if the automatic start is executed during MDI mode before not all axes have yet to complete automatic restart position return, the error "T01 Restart pos. return incomplete" occurs.
- (Note 6)** After restart search had been completed, if the movement command is issued by MDI before automatic restart position return has been started, the program error (P48) occurs. Perform the operation again after reset.
- (Note 7)** After restart search had been completed, if the T command different from the program is issued by MDI before the searched program has been started, the program error (P48) may occur. Perform the operation again after reset. T commands which result in errors are shown below. (Only L system)
- (1) When "#1100 Tmove" = "0"
T command that selects a tool with different tool length/wear amount than the tool selected by the program
 - (2) When "#1100 Tmove" = "1"
A program error will not occur by T command.
 - (3) When "#1100 Tmove" = "2"
T command that selects a tool with different wear amount than the tool selected by the program
- (Note 8)** After restart search had been completed, if the handle interruption is performed before the searched program has been started, always the same operation as manual absolute ON is executed regardless of "Manual absolute" signal. When manual interruption is executed to the automatic restart position return completed axis, the axis never returns the automatic restart position again.
- (Note 9)** When an axis moved to return is operated manually, the axis cannot return to correct position.

2.3.11 Executing the MSTB Commands

If the MSTB history menu is pressed after restart search is completed, the MSTB commands used for machining program appear.

When the cursor is moved to the listed M, S, T, B commands and the **INPUT** key is pressed, that command will be executed.

Up to 35 M commands, 3 commands each for S1 to S6, 3 T commands and 3 B commands are displayed. If many MSTB commands are used for machining, the MSTB commands used at first will not appear.

Operation method

- (1) Press the submenu **MSTB history**.  The pop-up window changes to the MSTB history window. The MSTB commands used for machining program are listed.
- (2) Using the , ,  and  keys, move the cursor to the position of the data to set.
- (3) Press the **INPUT** key.  The designated command is executed. A value, which has been commanded once, is displayed in gray. The cursor will move to the next item.
- (4) Repeat steps (2) and (3).
- (5) When completed with all settings, press the **Close** or **MSTB history** menu.  The MSTB history pop-up window closes and the restart search main window appears as a pop-up window.

2.4 Program Edit

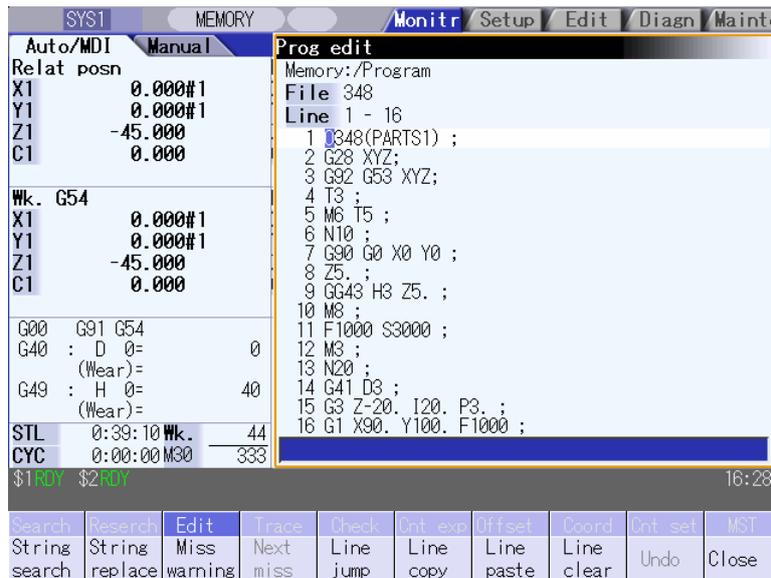


The machining programs are edited. When the main menu **Edit** is pressed, the operation searched program (MDI program for MDI mode) appears.

If no program has been searched or tape operation has been executed, the edit window will not open.

When the program is edited, the key input data is directly written into the program display area. All data is overwritten from the cursor position. "Editing" appears on the right side of the file name display when the input starts. When the **INPUT** key is pressed, the program is saved in the NC memory and the "Editing" message disappears.

Refer to "4.2 Program edit" for details.



2.5 Trace



This function illustrates the actual machine's movement path or tool center point movement path, and draws the actual machine movement.

This allows the machine operation to be monitored during machining.

The programs that can be traced are operation searched machining programs (MDI program for MDI mode).

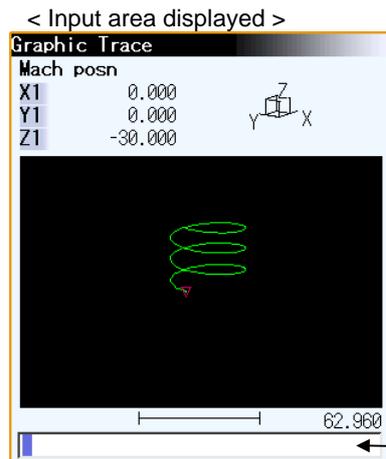
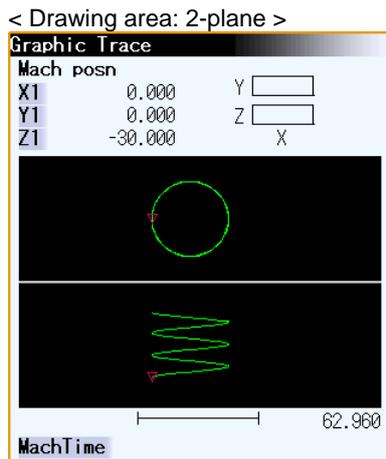
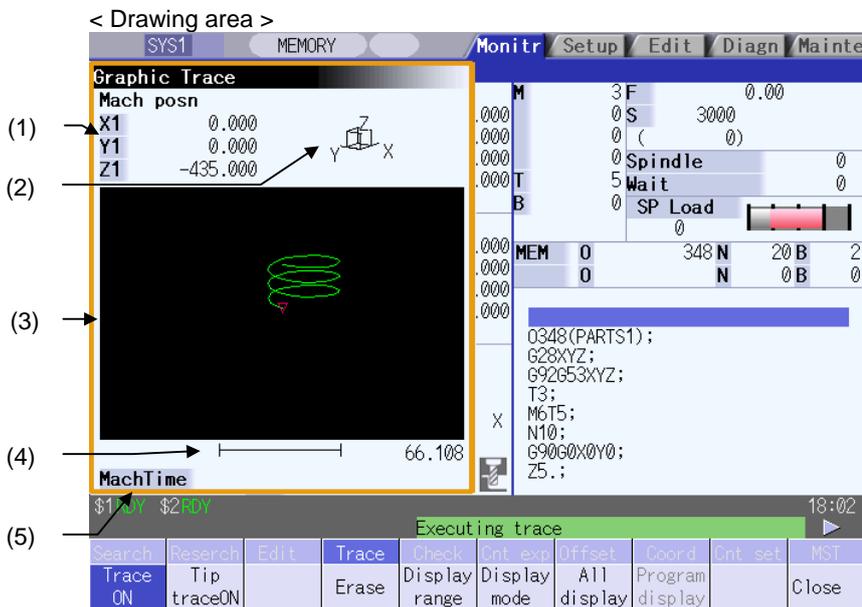
If no program has been operation searched, the trace window will not open.

Using **All display** menu, normal display and whole display modes can be switched.

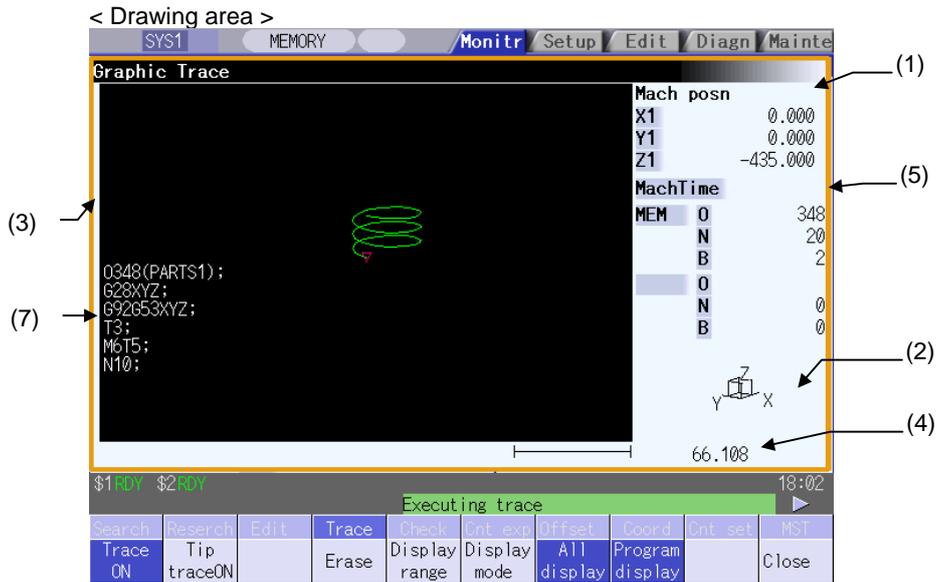
(Note 1) The trace function is an additional specification. The graphic trace option is required.

(Note 2) The tool center point trace function is an additional specifications. The 5-axis related options (tool center point control, tool length compensation along the tool axis, tool handle feed & interrupt) are required.

■ Normal display (full-screen display mode is OFF)



■ Full-screen display (full-screen display mode is ON)



Display items

Display item	Details
(1) Counter	The counter of the axis targeted for the trace drawing is displayed. The three axes which are displayed are set with the parameters. The counter selected by the menu "Counter type" is displayed. The counter type displayed can be set for [Trace ON] and [Tip traceON] separately. And the setting is held even after the power is turned OFF and ON. The default setting is as follows. [Trace ON] : depends on the setting of "#1231 set03 (bit4)". (0: The machine position counter (Mach posn), 1: The workpiece coordinate position counter (Work co posn)) [Tip traceON] : The machine position of the tool center point counter (Tip Mach posn)
(2) Display mode	The plane currently being drawn is displayed.
(3) Trace drawing area	This area is used to draw the tool path as graphics. The zero point to be displayed is as follows. Machine position trace mode : The zero point can be switched between the machine position zero point and workpiece coordinate position zero point with parameter "#1231 set03". Tool center point trace mode : The machine position of tool center point zero point
(4) Scale	The display range scale is displayed.
(5) Machining time display	The time required for machining is calculated and displayed. Nothing is displayed during tracing.
(6) Input area	The input area appears when the menu <input type="button" value="Display range"/> and <input type="button" value="Display mode"/> are pressed. The scale value and display mode are set. The input area is hidden when the <input type="button" value="INPUT"/> key is pressed.
(7) Program display	This displays the contents of the machining program currently being executed. The block being executed is highlighted.

Menus

Menu	Details	Type	Reference
Trace ON	This activates the trace mode. If any program is currently running, the machine position path is traced from the current position. If this menu is pressed during the machine position trace mode, the trace mode will be turned OFF.	B	2.5.1 Displaying the Machine Position Trace
Tip traceON	This activates the tool center point trace mode. If any program is currently running, the path of tool center point machine position path is traced from the current position. If this menu is pressed during the tool center point trace mode, the trace mode will be turned OFF. Note that this menu does not appear if the 5-axis related option is OFF.	B	2.5.3 Displaying the Tool Center Point Trace
Erase	This erases the data in graphic drawing area displayed on the screen.	C	
Display range	This changes the graphic drawing display range. When this menu is pressed, the menu changes to the display range change menu. When the display range is changed, the graphic data displayed on the screen is erased.	C	2.5.5 Changing the Display Range
Display mode	This changes the drawing plane. When this menu is pressed, the menu changes to the display mode change menu. There are three types of graphic display modes: 1-plane, 2-plane and 3D. When the drawing plane is changed, the graphic data displayed on the screen is erased.	C	2.5.6 Changing the Display Mode
All display	This switches the normal display mode and the full-screen display mode.	B	2.5.8 Switching the Full-screen Display Mode
Program display	This displays the machining program being executed on the graphic drawing area. This menu can be selected only when the full-screen display mode is applied.	B	
Rotate	This sets the viewpoint angle for the 3D display mode. When the viewpoint angle is changed, the graphic data displayed on the screen is erased. Note that this menu can be used only in the 3D display mode.	C	2.5.7 Changing the Display Angle
Std range	The display range (scale and display position) is automatically set from the machine movable area. The machine movable area is set with the parameters "#2013 OT-" and "2014 OT+" (software limit). When the display range is changed, the graphic data displayed on the screen is erased.	C	
Counter type	This displays the selected counter. For each trace mode, the counter type can be set separately. And the setting is held after the power is turned OFF and ON. However, if the [Trace ON] or [Tip traceON] menu is not highlighted, this is displayed in gray and cannot be selected. The following counters can be selected. <ul style="list-style-type: none"> • Relat posn (Related position counter) • Work co posn (Workpiece coordinate position counter) • Machine posn (Machine position counter) • Remain command (Remain command counter) • Next command (Next command counter) • Program posn (Program position counter) • Tip wk posn (Tool tip workpiece position counter) (Note) • Tip machine (Machine position of tool center point counter) (Note) • Pulse (Tool axis movement counter) (Note) (Note) These counters can be selected only when the <input type="button" value="Tip traceON"/> menu is highlighted.	C	
Close	This closes the pop-up window and quits this function.	C	

2.5.1 Displaying the Machine Position Trace

Operation method

(1) Press the main menu .

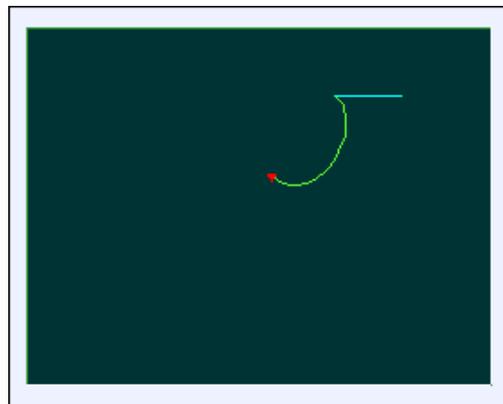


- The menu is highlighted.
- If is highlighted, unhighlight it.
- The machine position appears in the drawing area as a tool mark.
- The counter selected by menu is displayed at the counter.
- The message "Tracing" appears.

After this, the machine position path is drawn with graphics in the trace mode.



The path where the machine position or tool center position is drawn with graphics.



The path is drawn with a solid green line.

2.5.2 Canceling the Machine Position Trace

Operation method

- (1) Start tracing with the machine position.
- (2) Press the main menu **Trace ON**.
- The **Trace ON** menu is unhighlighted.
 - The tool mark in the drawing area is erased.
 - The message "Tracing" is erased.
- (Note)** Even if the menu is unhighlighted, the displayed counter type does not change.

2.5.3 Displaying the Machine Position of the Tool Center Point Trace

Operation method

- (1) Press the main menu **Tip TraceON**.
- The **Tip TraceON** menu is highlighted.
 - If **Trace ON** is highlighted, unhighlight it.
 - The tool center point appears in the drawing area as a tool mark.
 - The counter selected by **Counter type** menu is displayed at the counter.
 - The message "Tracing Tip Position" appears.

After this, the tool center point path is drawn with graphics in the tool center point trace mode.

2.5.4 Canceling the Machine Position of the Tool Center Point Trace

Operation method

- (1) Start tracing with the tool center point.
- (2) Press the main menu **Tip TraceON**.
- The **Tip TraceON** menu is unhighlighted.
 - The tool mark in the drawing area is erased.
 - The message "Tracing Tip Position" is erased.
- (Note)** Even if the menu is unhighlighted, the displayed counter type does not change.

2.5.5 Changing the Display Range

The graphic drawing's scale can be enlarged or reduced, and the position moved or centered.

Operation method (Enlarging the and reducing the drawing)

- (1) Press the key. ➔ A white frame indicating the display range appears on the screen.
The display range input mode is activated, and the following menu appears.

Zoom in	Zoom out	Up	Down	Left	Right	Centering			
---------	----------	----	------	------	-------	-----------	--	--	--

- (2) Press the menu or . ➔ The size of the white frame changes according to the key operations.

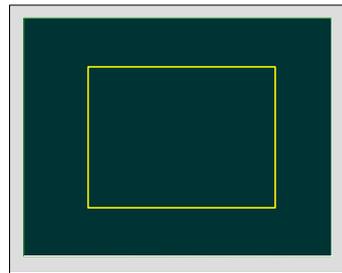
To enlarge the figure:

Press the menu or the key.

To reduce the figure:

Press the menu or the key.

A solid-line frame appears with respect to the original scale when enlarging, and a dotted-line frame appears when reducing.



- (3) Press the key. ➔ The display scale is changed.
By changing the display scale, the graphic data displayed on the screen is erased.

(Note) The changed scale value is held even after the power is turned OFF and ON.

Operation method (Changing the drawing display position)

(1) Press the key.



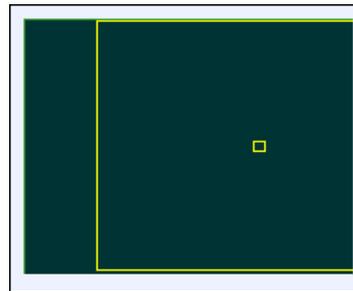
A white frame indicating the display range appears on the screen.
The display range input mode is activated, and the following menu appears.

Zoom in	Zoom out	Up	Down	Left	Right	Centering			
---------	----------	----	------	------	-------	-----------	--	--	--

(2) Press the , , , menu or cursor movement keys , , , .



The cursor () indicating the center of the display and the frame line move up, down, left and right according to the key operations.

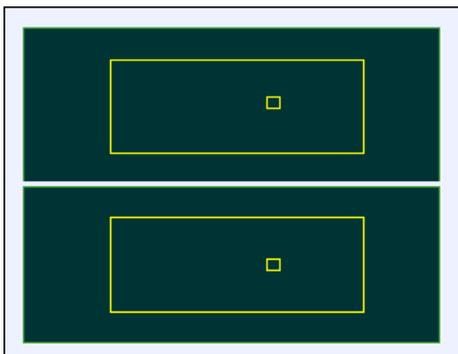


(3) Press the key.



The display position changes so that the cursor position () is the center of the area.
By changing the display position, the graphic data displayed on the screen is erased, however, the scale value is not changed.

(Note 1) When changing the display range in the 2-plane display mode such as "XY/XZ", the display range (scale and display position) for the upper and lower areas changes in the same manner. The operation method is the same as the 1-plane display mode.



The white-framed display range in the upper and lower areas simultaneously move to the left/right when the , keys are pressed.

The white-framed display range of either upper or lower areas moves up and down when the , keys are pressed.

Which frame to be moved up/down can be changed with the page key.

Operation method (Centering)

- (1) Press the key. ➔ A white frame indicating the display range appears on the screen.
The display range input mode is activated, and the following menu appears.

Zoom in	Zoom out	Up	Down	Left	Right	Centering			
---------	----------	----	------	------	-------	-----------	--	--	--

- (2) Press the key. ➔ The display position is changed so that the current machine position appears in the center of the drawing area.
When display position is changed, the graphic data displayed on the screen is erased.

(Note 1) In the 2-plane display mode such as "XY/XZ", centering is applied for the upper and lower areas. The operation method is the same as the 1-plane display mode.

2.5.6 Changing the Display Mode

The graphics display mode includes the 1-plane, 2-plane and 3D mode. When the menu is pressed and the following display mode menu is selected, the axis configuration of each plane changes, and the menu display returns to the original state. It is also possible to change the display mode by setting the axis name to the input area.

(Note 1) When display mode is changed, the graphic data displayed up to that point are erased.

(Note 2) The set display mode is held even after the power is turned OFF and ON.

(Note 3) The display mode can be set independently for the trace function and program check function. These settings are not sequenced.

(Note 4) The menu axis names XYZ correspond to the base axis IJK. X = base axis I, Y = base axis J, and Z = base axis K.

Display mode menus

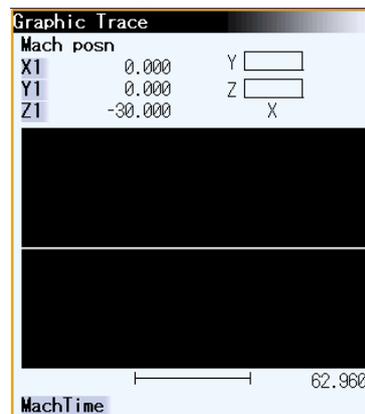
Menu	Details	Type
<input type="text" value="XY"/>	This changes to the 1-plane display mode configured of X-Y. The X axis is displayed as the horizontal axis, and the Y axis is displayed as the vertical axis.	C
<input type="text" value="YZ"/>	This changes to the 1-plane display mode configured of Y-Z. The Y axis is displayed as the horizontal axis, and the Z axis is displayed as the vertical axis.	C
<input type="text" value="XZ"/>	This changes to the 1-plane display mode configured of X-Z. The X axis is displayed as the horizontal axis, and the Z axis is displayed as the vertical axis.	C
<input type="text" value="XY/XZ"/>	This changes to the 2-plane display mode configured of X-Y and X-Z. The X axis is displayed as the horizontal axis, and the Y axis and Z axis are displayed as the vertical axis.	C
<input type="text" value="YX/YZ"/>	This changes to the 2-plane display mode configured of Y-X and Y-Z. The Y axis is displayed as the horizontal axis, and the X axis and Z axis are displayed as the vertical axis.	C
<input type="text" value="XYZ"/>	This changes to the 3D display mode. A cube is displayed on the lower right of the screen.	C

Operation method (Changing the display mode)

(1) Press the menu . → The menus for the selectable display modes appear. The input area appears.

(2) Select the mode to display and press the menu. → The display mode and drawing area appear as a 2-plane image.

(Example)



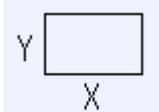
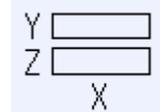
Operation method (Changing the display axis name)

- (1) Press the menu . → The menus for the selectable display modes appear. The input area appears.
- (2) Set the axis name, and press the key. → XYC appears as the axis name in the display mode. The X axis, Y axis and C axis are the three axes for drawing.

XYC

The display mode includes the 1-plane, 2-plane and 3D display modes. The display mode currently selected is shown with an image. The selected axis names are displayed in the image.

Display example

<1-plane> (XY)	<2-plane> (XY/XZ)	<3D> (XYZ)
		

2.5.7 Changing the Display Angle

Set the graphics display angle for the 3D display mode.

Press the menu key or the cursor movement keys, and rotate the cube displayed at the upper right of the screen. Set the angle by pressing the key.

(Note 1) When the display angle is changed, the graphics displayed up to that point are erased.

(Note 2) The set display angle is held even if the power is turned ON and OFF.

(Note 3) The display angle can be set independently for the trace function and program check function. These settings are not sequenced.

Operation method

- (1) Press the menu key . → The rotation mode is activated, and the following menu appears.

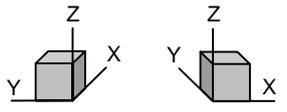
Up

Down

Left

Right
- (2) Use the , , , menu to adjust the display angle. → The cube displayed at the lower right of the screen rotates according to the key operations.

The cursor movement keys (, , ,) can also be used to rotate.


- (3) Press the key. → The display angle is changed. When display angle is changed, the graphic data displayed on the screen is erased.

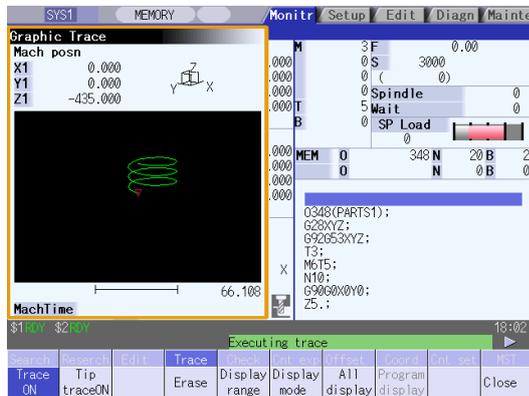
(Note 1) The set display angle is held even after the power is turned OFF and ON.

2.5.8 Switching the Full-screen Display Mode

Press the **All display** menu key to display the trace window on the whole screen. (full-screen display mode).

Press this menu again to return the normal display.

Switching the display mode

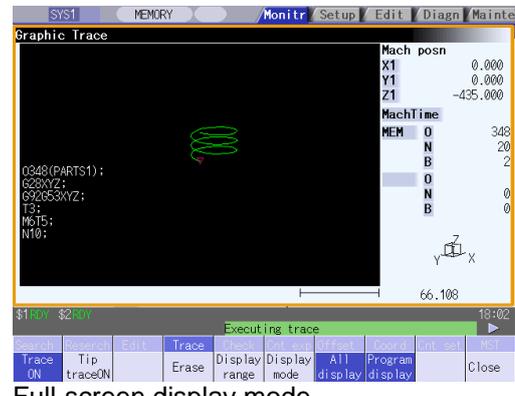


Normal display mode

All display
menu : ON



All display
menu : OFF



Full-screen display mode

(Note 1) The selected display mode is maintained even after rebooting the machine.

(Note 2) The full-screen display mode is applied to the trace function and program check (2D/3D solid) function commonly.

(Note 3) The full-screen display mode is applied commonly to all part systems.

Displaying a program

When pressing the **Program display** menu key, the contents of the machining program currently being checked is displayed on the drawing area.

When pressing the **Program display** again, the displayed program is erased.

Note that this menu can be used only when the full-screen display mode is selected.

(Note 1) The selected display mode is maintained even after rebooting the machine.

(Note 2) The program display mode is applied to the trace function and program check (2D) function commonly.

(Note 3) The **Program display** menu key cannot be selected while the normal display mode is selected. Program display mode is valid only when the full-screen display mode is selected.

(Note 4) The program display mode is applied commonly to all part systems.

2.5.9 Precautions for Tracing

(1) When the command speed increases in the trace mode, the shape being drawn with graphics will become distorted. Especially, correct graphic drawing is not possible during machine lock.

(2) If program check (2D) is executed during the trace mode, the trace mode will be canceled.

(3) If trace is executed during program check mode, the program check will be canceled.

Refer to section "4.3.11 Precautions for Program Check Function (2D)" for other precautions.

2.6 Program Check (2D)



Program check (2D) is a function that draws the machining program movement path without executing automatic operation. The machining program can be checked with graphic data drawn at a high speed. Using All display menu, normal display and full-screen display modes can be switched.

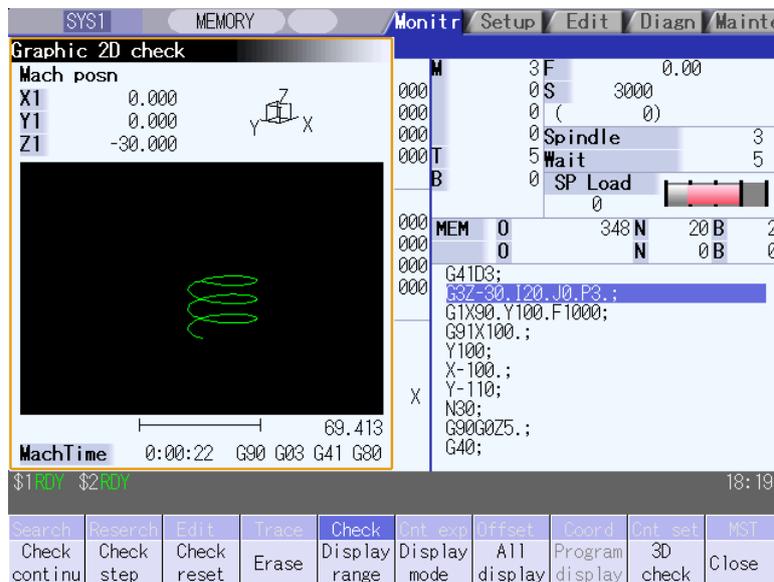
The programs that can be checked are operation searched machining programs (MDI program for MDI mode).

If no machining program has been operation searched, the program check window will not be opened. Refer to "4.3 Program Check (2D)" for details.

(Note) The program check (2D) function is an additional specification. The graphic check option is required.

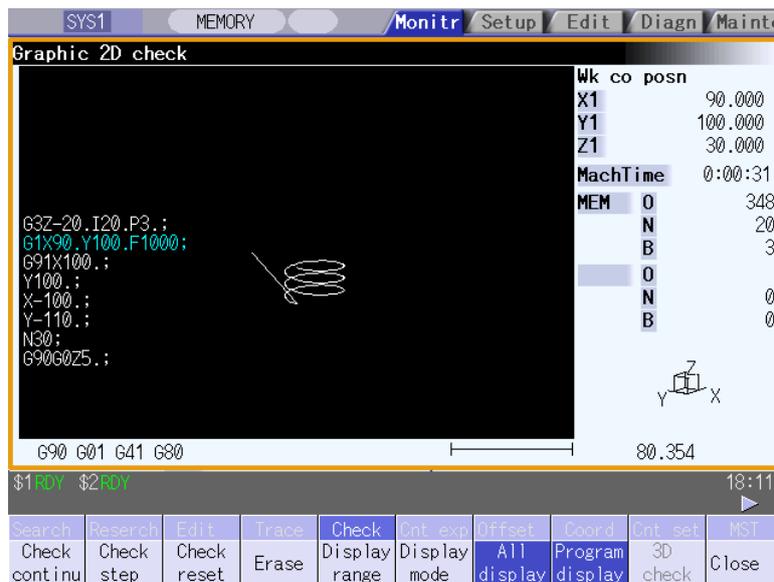
■ Normal display (full-screen display mode is OFF)

<Drawing area>



■ Full-screen display (full-screen display mode is ON)

<Drawing area >



2.7 Program Check (3D) [700 series only]



Program check (3D) is a function that draws the workpiece shape and tool movement in the cutting process of the machining program as a solid image without executing automatic operation. The machining program can be checked with graphic data drawn at a high speed.

Using All display menu, normal display and full-screen display modes can be switched.

The operation searched machining program (MDI program for MDI mode) can be checked.

The Program Check window will not open if no machining program has been searched.

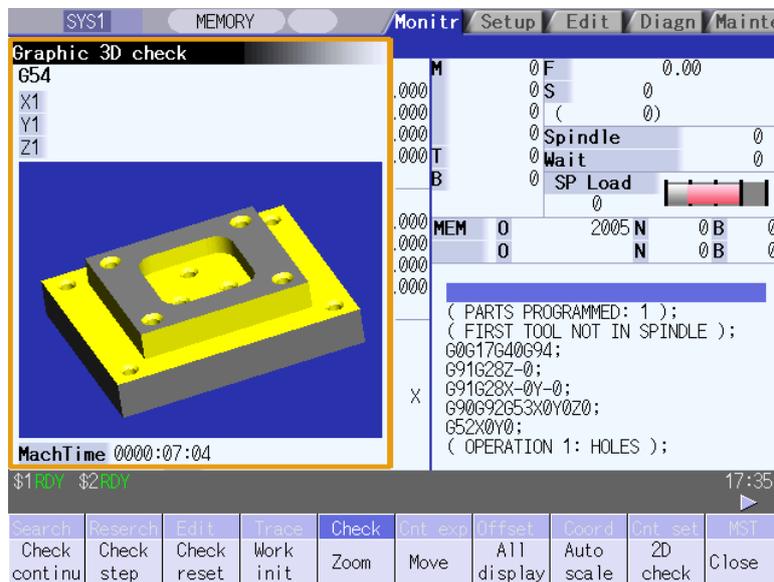
The workpiece shape and tool shape used with this function can be set on the Edit screen.

Refer to section "4.4 Program Check (3D)" for details.

(Note) The program check (3D) function is an additional specification. The graphic check and 3D solid graphic check options are required.

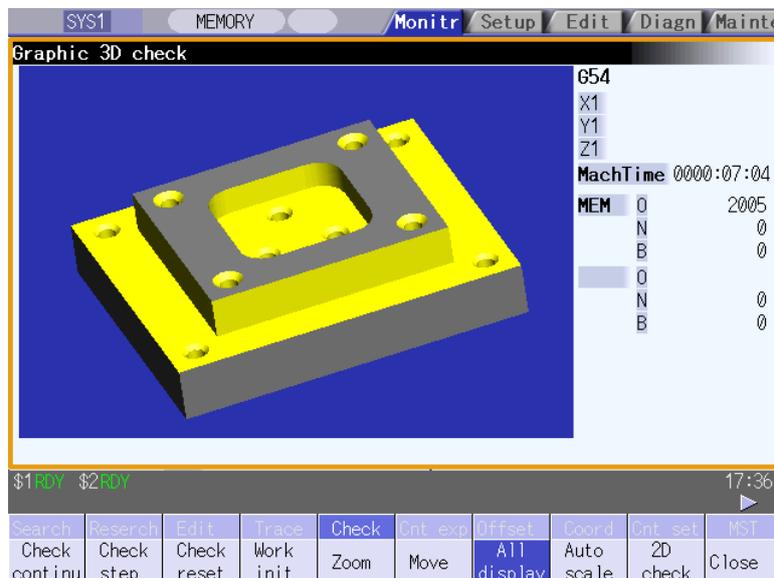
- Normal display (full-screen display mode is OFF)

<Drawing area>



- Full-screen display (full-screen display mode is ON)

<Drawing area>



2.8 Counter All-axis Display



A counter for all axes opens as a pop-up display.
 The type of displayed counter can be selected with the menu.

The screenshot shows the CNC control interface with a 'Counter displays' pop-up window. The window lists relative positions for axes X1, Y1, Z1, and C1. The background interface shows various machine parameters and a menu bar.

Axis	Counter Value
X1	0.000#1
Y1	0.000#1
Z1	55.000
C1	0.000

Background interface details:

- Menu bar: SYS1, MEMORY, Monitor, Setup, Edit, Diagn, Mainte
- Parameters: M 6 F 0.00, S 3000, Spindle 3, Wait 5, SP Load 0
- MEM: 0 348 N 10 B 3, 0 N 0 B 0
- Program: Z5.; G43H3Z5.;
- Machine: M8; F1000S3000; M3; N20; G41D3; G3Z-30.I20.J0.P3.; G1X90.Y100.F1000; G91X100.;
- Status: \$1BST \$2 Cnt exp 18:35
- Bottom menu: Search, Reserch, Edit, Trace, Check, Cnt exp, Offset, Coord, Unt set, MST; Relat, Work, Machine, Remain, Next, Manual, Program, Close

Menus

Menu	Details	Type
Relat posn	This displays the currently executed position	B
Work co posn	This displays the G54 to G59 workpiece coordinate system modal No. and the workpiece coordinate position in that workpiece coordinate system.	B
Machine posn	This displays the coordinate position of each axis in the basic machine coordinate system having a characteristic position, specified by the machine, as a zero point.	B
Remain command	This displays the remaining distance of the movement command being executed during automatic start or automatic halt. (The remaining distance is the incremental distance from the current position to the end point of that block.)	B
Next command	This displays the details of the command in the block executed after the block currently being executed.	B
Manual int amt	This displays the amount moved with the manual mode while the manual absolute switch was OFF.	B
Program posn	This displays the value obtained by subtracting the tool compensation amount compensated for that axis from the position actually being executed for each axis.	B
Close	This closes the pop-up window and quits this function.	C
Tip wk posn	This displays the position of the tool end from the workpiece coordinate reference position in the selected workpiece coordinate system.	B
Tip machine	This displays the position of the tool end from the machine coordinate system reference position in the machine coordinate system.	B
Pulse	This displays the amount moved in the selected axis direction using the manual pulse generator in the hypothetical machine coordinate system. Basically this is updated only when manual ABS is OFF. If "#7905 NO_ABS" is set to "1", this will be updated regardless of the manual ABS ON/OFF.	B

(Note 1) The type of counter displayed first when the pop-up display appears is the relative position. If the 5-axis related option is ON, the counter will be the "Tip wk posn".
The counter which appears next is the type selected previously.

(Note 2) The menus Tip wk posn, Tip machine and Tip axis movement appear when the 5-axis related option is ON.

2.9 Tool Compensation Amount



The tool compensation data can be set and displayed.
 The tool compensation data screen configuration differs according to the tool compensation type.
 The number of tool compensation sets to be set or shown differs according to the option.
 Refer to section "3.2 Tool Compensation Amount" for details.

[Tool compensation type I (M system)] Parameter "#1037 cmdtyp" = 1

The combined amount of the shape compensation and wear compensation are set as the compensation data, with no distinction between shape compensation memory and wear compensation memory. (The tool compensation data is the shape compensation amount + wear compensation amount.)
 The compensation data is used commonly for the tool length compensation, tool position offset, tool radius compensation, and three-dimensional tool radius compensation.

Tool No.	Compensation	Tool ID	Compensation	Tool ID	Compensation
1	0.000	16	0.000	31	0.000
2	0.000	17	0.000	32	0.000
3	-500.000	18	0.000	33	0.000
4	0.000	19	0.000	34	0.000
5	0.000	20	0.000	35	0.000
6	0.000	21	0.000	36	0.000
7	0.000	22	0.000	37	0.000
8	0.000	23	0.000	38	0.000
9	0.000	24	0.000	39	0.000
10	0.000	25	0.000	40	0.000
11	0.000	26	0.000	41	0.000
12	0.000	27	0.000	42	0.000
13	0.000	28	0.000	43	0.000
14	0.000	29	0.000	44	0.000
15	0.000	30	0.000	45	0.000

[Tool compensation type II (M system)] Parameter "#1037 cmdtyp" = 2

The shape compensation amount and wear compensation amount are set separately. The shape compensation amount is furthermore divided into length and radius dimensions.
 Of the compensation data, the length dimension data is used for the tool length compensation and tool position offset, and the radius dimension data is used for the tool radius compensation and three-dimensional tool radius compensation.

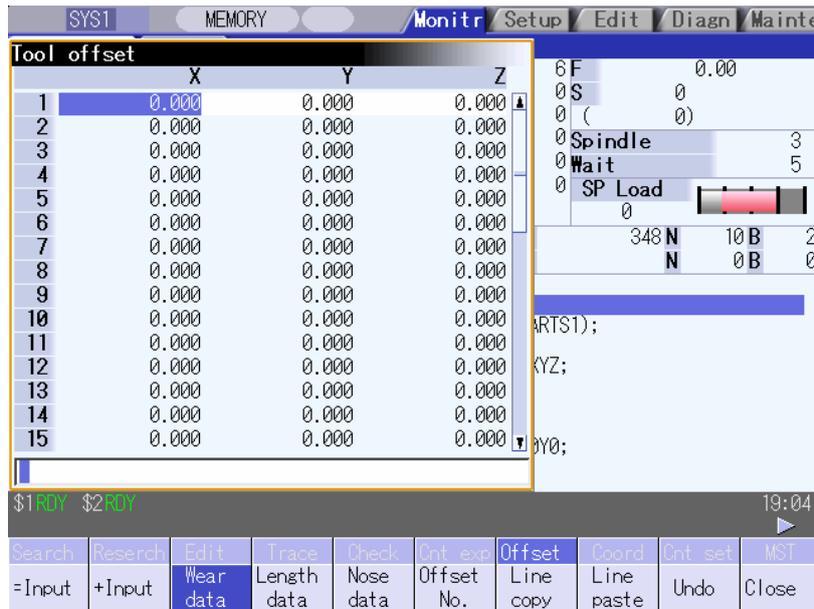
Tool No.	Length	L wear	Radius	R wear
1	0.000	0.000	10.000	0.000
2	0.000	0.000	0.000	0.000
3	-500.000	0.000	0.000	0.000
4	0.000	0.000	0.000	0.000
5	0.000	0.000	0.000	0.000
6	0.000	0.000	0.000	0.000
7	0.000	0.000	0.000	0.000
8	0.000	0.000	0.000	0.000
9	0.000	0.000	0.000	0.000
10	0.000	0.000	0.000	0.000
11	0.000	0.000	0.000	0.000
12	0.000	0.000	0.000	0.000
13	0.000	0.000	0.000	0.000
14	0.000	0.000	0.000	0.000
15	0.000	0.000	0.000	0.000

[Tool compensation type III (L system)] Parameter "#1037 cmdtyp" = 3

The wear data, tool length data and tool nose data are set separately. These are changed with the sub-menu.

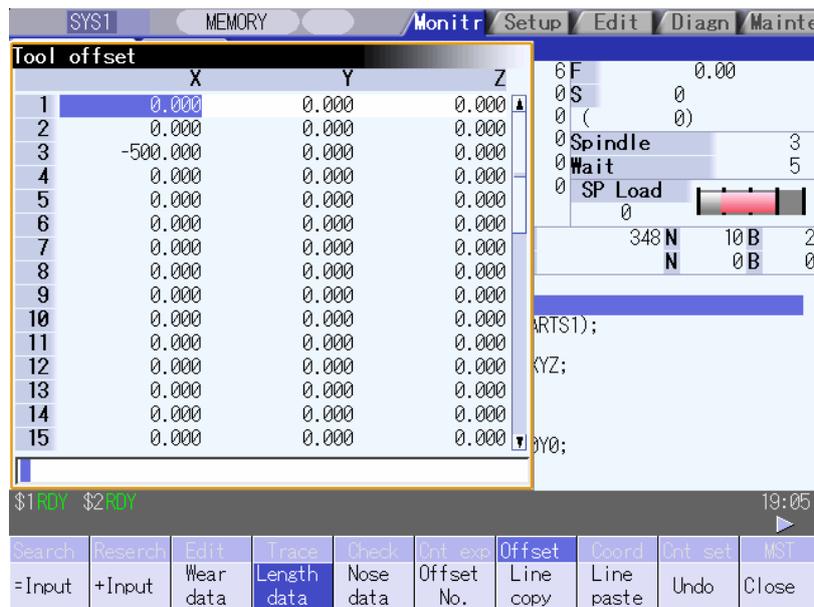
(a) Wear data

Set the tool nose wear amount for each tool used. When the tool compensation No. is designated by the tool command (T command), compensation is carried out matching the tool length data and tool nose data.



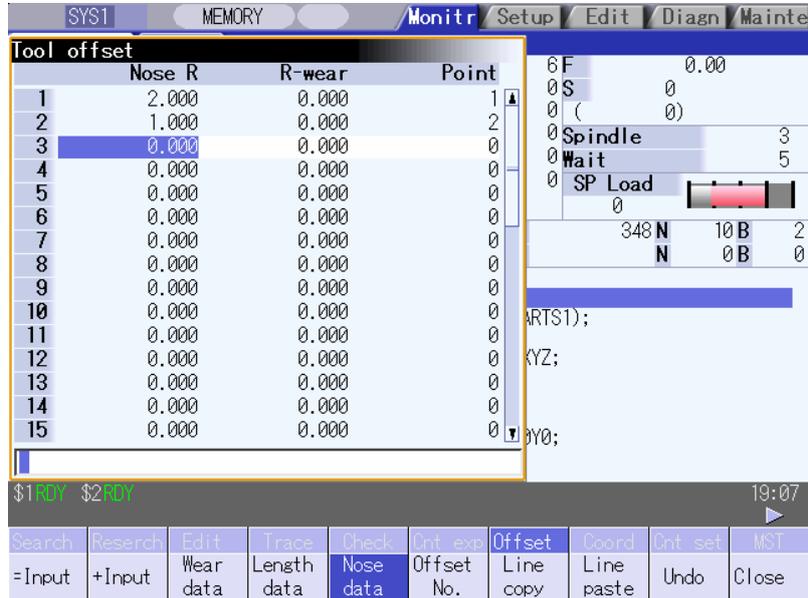
(b) Tool length data

Set the tool length in respect to the program basic position of each tool used. When the tool compensation No. is designated by the tool command (T command), compensation is carried out matching the wear data and the tool nose data.



(c) Tool nose data

Set the tool nose radius value (tool nose R), wear radius value (R wear) and tool nose point (tool nose point P) of the tool nose mounted on the tool for each tool used. When the tool offset No. is designated by the tool command (T command), offset is carried out matching the tool length data and tool nose data.



CAUTION

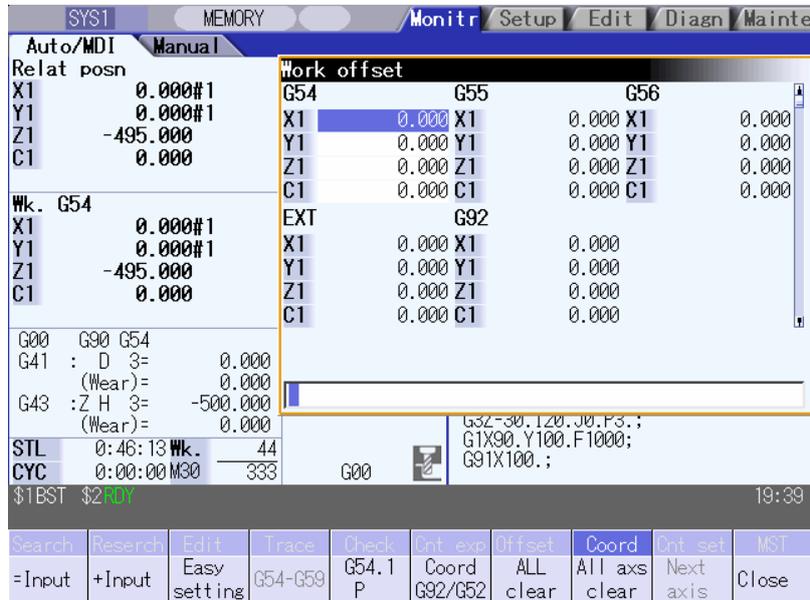
If the tool offset amount or workpiece coordinate system offset amount is changed during automatic operation (including during single block stop), the changes will be valid from the command in the next block or after several subsequent blocks.

2.10 Workpiece Coordinate System Compensation



The coordinate system offset controlled by the NC can be set and displayed. 48 or 96 sets of coordinate system offset sets can be added according to the option. Refer to "3.6 Workpiece Coordinate System Offset" for details.

(Note) The G92/G52 coordinate system offset cannot be set.



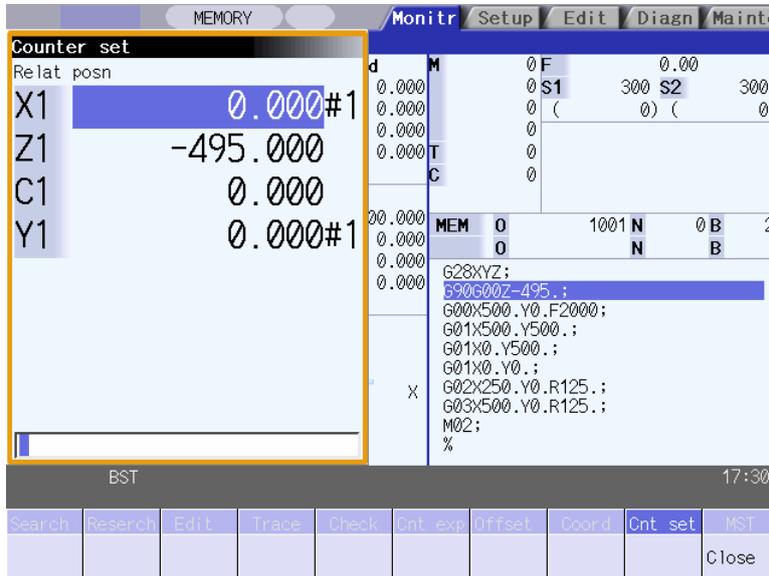
CAUTION

If the tool offset amount or workpiece coordinate system offset amount is changed during automatic operation (including during single block stop), the changes will be valid from the command in the next block or after several subsequent blocks.

2.11 Counter Set



An arbitrary value can be set in the relative position counter which opens as a pop-up window. The counter cannot be set for an axis of auxiliary axis state.



Menus

Menu	Details	Type	Reference
Close	This closes the pop-up window and quits this function.	C	

Operation method

- (1) Press the main menu . → The relative position counter opens as a pop-up window. The cursor appears at the 1st axis of the relative position counter.

Counter set	
Relat posn	
X1	0.000#1
Z1	-495.000
C1	0.000
Y1	0.000#1

Instead of the operation above, the axis name address key can be pressed to set the counter.

(Example)

- The relative position counter opens as a pop-up window. The cursor appears at the designated axis (Z axis).

Counter set	
Relat posn	
X1	0.000#1
Z1	-495.000
C1	0.000
Y1	0.000#1

- (2) Using the , , or keys, move the cursor to the axis to be set. → The cursor moves.

(Note) If two axes or more of the same name exist such as mixed synchronization control, the cursor moves to the axis corresponding to the address key to the axis name input first.

- (3) Input a numeric value. → The set value "100.000" appears at the cursor position. The cursor moves to the next axis.

(Example) 100.000

If the key is pressed without inputting a value, zero (0) will be set.

Counter set	
Relat posn	
X1	100.000#1
Z1	-495.000
C1	0.000
Y1	0.000#1

- (4) Repeat step (3) for each axis. → The pop-up window closes when the counter has been set for the last axis.

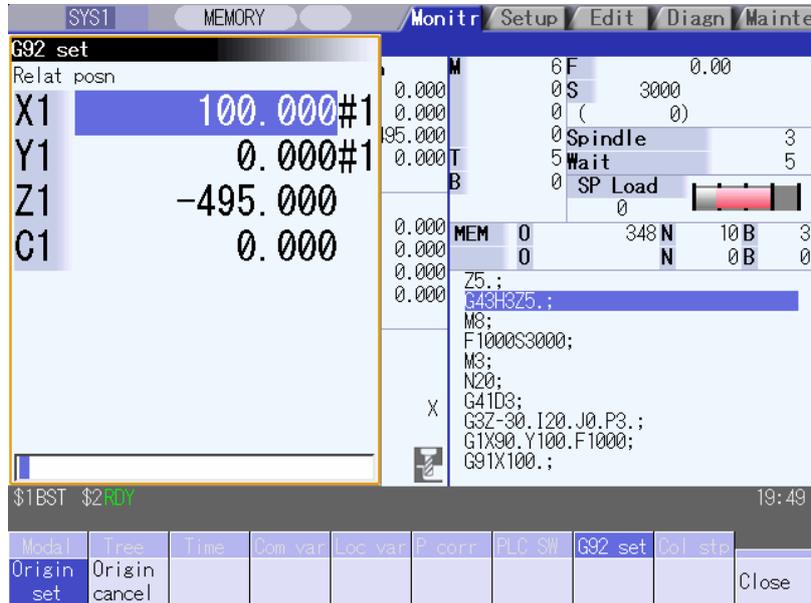
For axes that are not to be operated, press the key and move the cursor.

2.12 Origin Set, Origin Cancel



Origin set and origin cancel can be executed.

Origin set and origin cancel cannot be executed for an axis of auxiliary axis state.



Menus

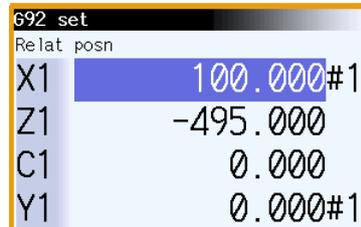
Menu	Details	Type	Reference
Origin set	The origin is set. When main menu G92 set is pressed, the menu Origin set is highlighted and the origin set mode is activated.	A	
Origin cancel	The origin is canceled. The origin cancel mode is activated when this menu is pressed.	A	
Close	This closes the pop-up window and quits this function.	C	

Differences between origin set and origin cancel

	Origin set	Origin cancel
Function	The coordinate system is shifted so the current position becomes the designated position of the workpiece coordinate system. This is equivalent to "G92 X0". (When X is the target axis.)	The coordinate system is shifted so the zero point of the workpiece coordinate system matches the zero point of the basic machine coordinate system. This is equivalent to "G92 G53 X0". (When X is the target axis.)
Changes in the counter value	The relative position counter and workpiece coordinate counters become "0".	The relative position counter matches the machine coordinate counter.
Offset	The G92 shift amount is updated.	The G92 shift amount and the G52 shift amount for the G54 to G59 coordinate system becomes "0".

Operation method (Setting the origin)

- (1) Press the main menu . → The relative position counter opens as a pop-up window.
The menu is highlighted.
The cursor appears at the 1st axis of the relative position counter.



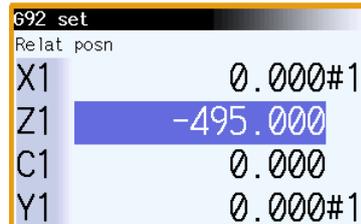
```

G92 set
Relat posn
X1 100.000#1
Z1 -495.000
C1 0.000
Y1 0.000#1
  
```

- (2) Using the , , or keys, move the cursor to the axis to be set. → The cursor moves.

(Note) If two axes or more of the same name exist such as mixed synchronization control, the cursor moves to the axis corresponding to the address key to the axis name input first.

- (3) Input 0, and then press the key. → "0.000" appears at the cursor position.
The cursor moves to the next axis.
If the key is pressed without inputting a value, zero (0) will be set.
(Origin zero)



```

G92 set
Relat posn
X1 0.000#1
Z1 -495.000
C1 0.000
Y1 0.000#1
  
```

- (4) Repeat step (3) for each axis. → The pop-up window closes when the origin has been set for the last axis.
For axes that are not to be operated, press the key and move the cursor.

(Note) An error will occur if a value other than 0 is set.

Operation method (Canceling the origin)

- (1) Press the main menu , and then press the menu . → The relative position counter opens as a pop-up window.
The menu is highlighted.
The cursor appears at the 1st axis of the relative position counter.

- (2) Press the key. → The same value as the machine position appears at the cursor position.
The cursor moves to the next axis.

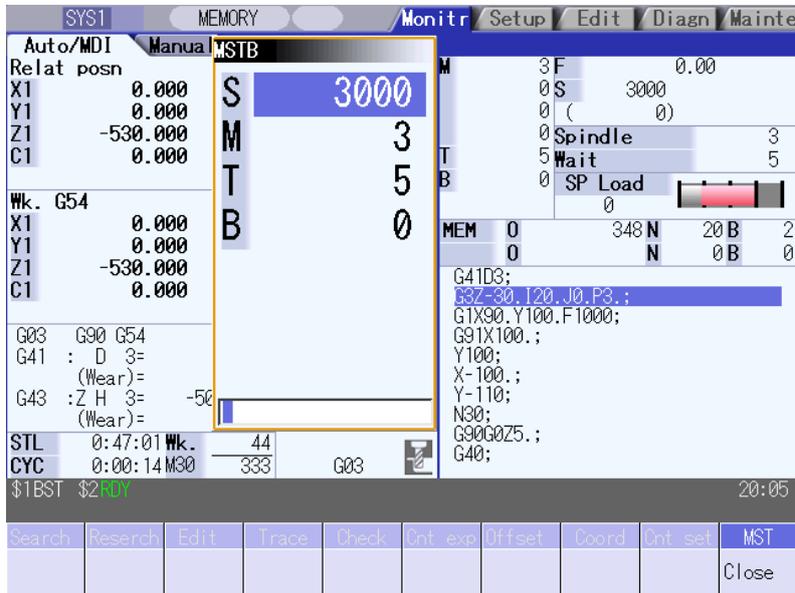
- (3) Repeat step (2) for each axis. → The pop-up window closes when the origin has been canceled for the last axis.
For axes that are not to be operated, press the key and move the cursor.

2.13 Manual Numerical Value Command



The spindle function (S), miscellaneous function (M), tool function (T) and 2nd miscellaneous function (B) commands can be executed.

The manual numerical value command can be executed by inputting an address such as S, M, T or B, as well.



Menus

Menu	Details	Type	Reference
Close	This closes the pop-up window and quits this function.	C	

Operation method (Executing T31 with a manual numerical value command)

- (1) Press the main menu → The S, M, T, B display opens as a pop-up window.
- | | |
|---|------|
| S | 100 |
| M | 50 |
| T | 4 |
| B | 1000 |
- Using the and keys, move the cursor to the position to be set. → The cursor moves.
- (2) The manual numerical value command can also be issued by using the address keys instead of the operation above. (Example) → The S, M, T, B display opens as a pop-up window. The cursor appears at T.
- | | |
|---|------|
| S | 100 |
| M | 50 |
| T | 4 |
| B | 1000 |
- (3) Input a value.
31 → The input value is set.

(Note) The cursor moves to the first line when the part system is changed while displaying the manual numeric command window.

Operation method (Canceling the manual numerical value command)

The manual numerical value command mode is canceled by carrying out one of the following operations before pressing the key.

- Press the menu
- Press the key
- Change the screen

Setting/output range of manual numerical value command

For each S, M, T and G command, the data type to output to PLC program is designated by parameters "#12006 Mbin M binary", "#12008 Sbin S binary", "#12010 Tbin T binary" and "#12012 Bbin B binary". The output data type and the manual numerical value command setting/output range are shown below.

		BCD	No-signed binary	Signed binary
Output range	S	-	0 to 99999999	-99999999 to 99999999
	M	0 to 99999999		
	T			
	B			

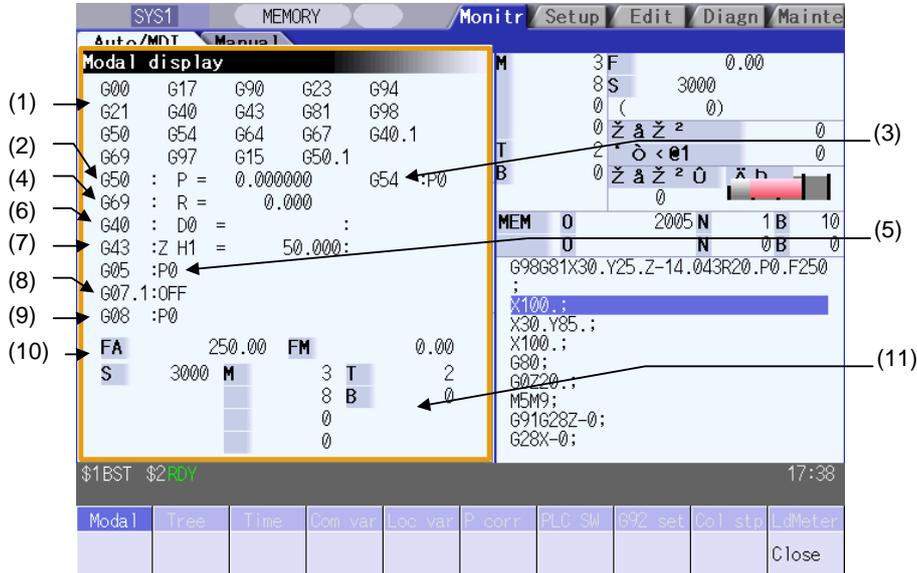
- (Note 1)** Even if the parameter (#12008 Sbin S binary) is set to "0" (BCD), the signed binary output is applied to S. BCD output is not applied.
- (Note 2)** If a negative value is set when the data type is "BCD type" or "No-signed binary type", a value converted into positive value is output to PLC program.
- (Note 3)** Add a "-" sign before the value to set a negative value. The display will be a positive value.
- (Note 4)** A value larger than "#3001 slimt1" to "#3004 slimt4" or "#3005 smax1" to "#3008 smax4" cannot be output for S.



2.14 Modal Display

The state of each modal during automatic operation is displayed.
The displayed details differ for the M system and L system.

<M system>

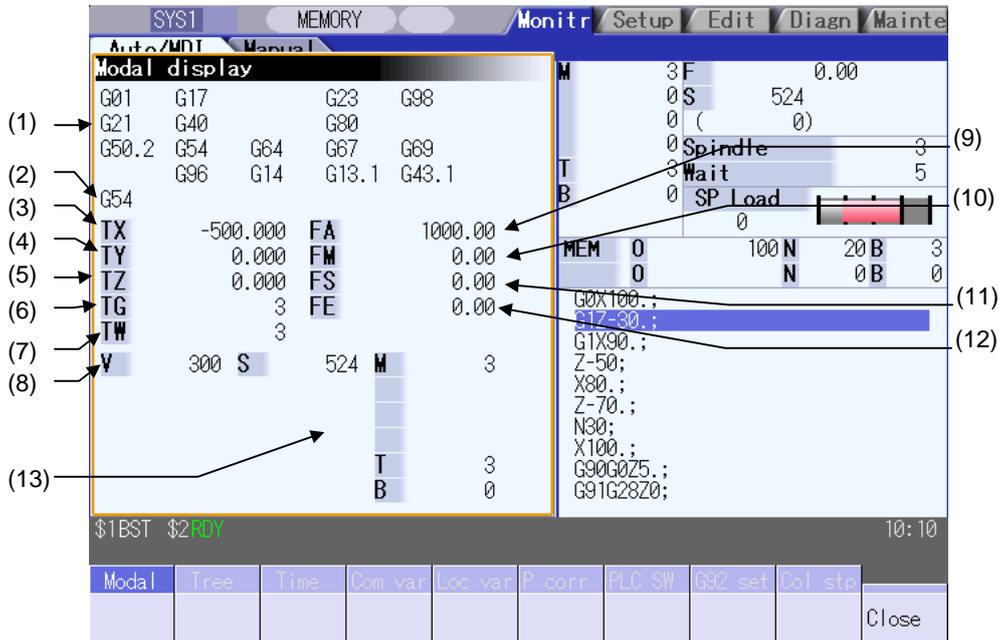


Display items

This displays each modal state.

Display item	Details
(1) G00...G94 : G69...G42.1	Status of currently executed G command modal
(2) G50:P=0.000000	Scaling magnification
(3) G54.1:P0	Extended workpiece coordinate system
(4) G69:R=0.000	Coordinate rotation angle (Displays within a ±360° range)
(5) G05:P0 Q1 Q2 P10000 P1 P2 P3 SSS	High-speed machining mode High-speed high-accuracy control 1 Spline interpolation High-speed high-accuracy control 2 (SSS control OFF) High-speed machining mode I High-speed machining mode II High-speed machining mode III SSS valid
(6) G40: D0 = 100.000000: 10.000000	Tool radius compensation modal Compensation No. Shape compensation amount in respect to tool radius Tool radius wear amount
(7) G49: Z H0 = 0.000000: 0.000000	Tool length compensation modal Compensation axis name Compensation No. Compensation amount Tool length wear amount
(8) G07.1:OFF	Cylindrical interpolation modal
(9) G08: P0 P1	High-accuracy control mode High-accuracy control mode OFF High-accuracy control mode ON
(10) FA FM	F modal value of currently executed program command Manual federate
(11) S1 to S4 M1 to M4 T B	Program command modal value of each currently executed command S command M command T command 2nd miscellaneous function (B) command

<L system>



Display items

This displays each modal state.

Display item	Details
(1) G01...G94 : G69...G42.1	Status of currently executed G command modal
(2) G54.1:P10	Extended workpiece coordinate system
(3) TX : -12.345	The total value of the first axis' tool length and wear compensation amount for the tool being used
(4) TZ : 12.345	The total value of the second axis' tool length and wear compensation amount for the tool being used
(5) TY : 10.000	The total value of the additional axis' tool length and wear compensation amount for the tool being used
(6) TG : 123	Tool length compensation No.
(7) TW: 123	Wear compensation No.
(8) V 12345678	Modal value for constant surface speed spindle rotation speed (V1 to V6)
(9) FA 24000.00	F modal value of currently executed program command
(10) FM 1200.00	Manual federate
(11) FS 0.0000	Program command synchronous feedrate modal value
(12) FE 0.0000	Thread lead command synchronous feedrate modal value
(13) S1 to S6 M1 to M4 T1 to T2 B	Program command modal value of each currently executed command S command M command T command 2nd miscellaneous function (B) command

(Note) Fixed cycle programs

When a fixed cycle command is executed, the G command in the fixed cycle subprogram is not affected by the G modal in the called program.

Menus

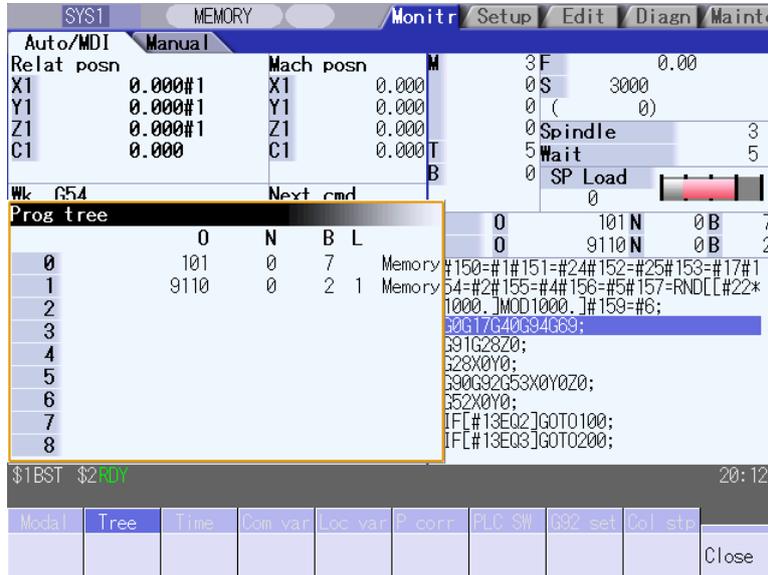
Menu	Details	Type	Reference
	This closes the pop-up window and quits this function.	C	

(Note) The menu is common for the M system and L system.

2.15 Program Tree Display



This displays the main program, subprogram, MDI interrupt and user macro call nesting structure.



Display items

Display item	Details
(1) 0	Program No. (0 to 15 characters) If the program No. (program name) exceeds 15 characters, "*" will appear at the 15th character.
(2) N	Sequence No. (0 to 99999)
(3) B	Block No. (0 to 99999)
(4) L	Remaining number of subprogram repetitions (0 to 99)
(5) Mode	Operation mode (0 to 7 characters)
(6) Main	Main program
(7) 1 to 8	Called subprogram, user macro call and MDI interrupt

Menus

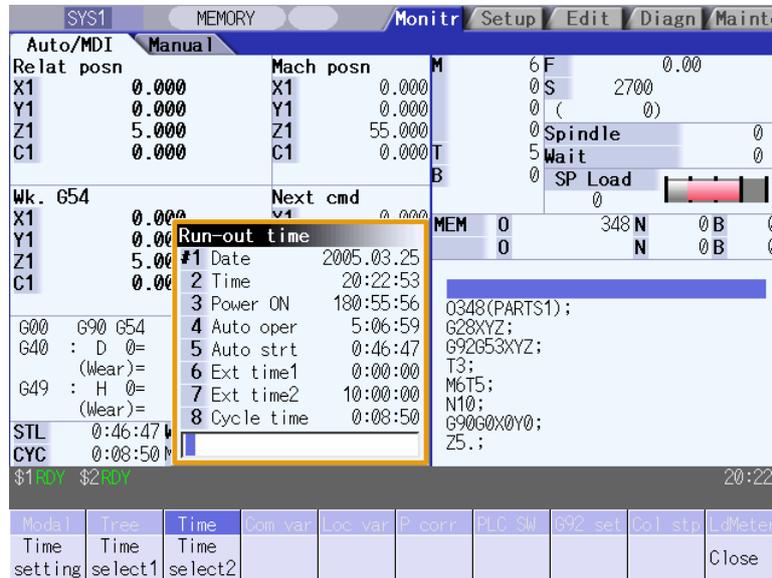
Menu	Details	Type	Reference
Close	This closes the pop-up window and quits this function.	C	

2.16 Integrated Time Display



The integrated time (date, time, power ON time, automatic operation time, automatic start time, external integrated time 1, external integrated time 2, cycle time) controlled by the NC can be set and displayed. Note that the cycle time cannot be set. (Cycle time is display only.)

The integrated times displayed in cycle time display area on "Auto/MDI" tag can be set.



Display items

Display item	Details
(1) #1 Date	The current date set in the NC is displayed. Year: 4 digits, Month: 2 digit, Date: 2 digit (YYYY.MM.DD)
(2) 2 Time	The current time set in the NC is displayed with the 24-hour system. (HH:MM:SS)
(3) 3 Power ON	This displays the total integrated time of the time from NC power ON to OFF. (HHHH:MM:SS)
(4) 4 Auto oper	This displays the total integrated time of the work time from automatic start button pressing in the memory (tape) mode to M02/M30 or reset button pressing (HHHH:MM:SS)
(5) 5 Auto strt	This displays the total integrated time during automatic starting from automatic start button pressing in the memory (tape) mode or MDI to feed hold stop, block stop, or reset button pressing. (HHHH:MM:SS)
(6) 6 Ext time1	This content differs depending on machine tool builder specification. (HHHH:MM:SS)
(7) 7 Ext time2	This content differs depending on machine tool builder specification. (HHHH:MM:SS)
(8) 8 Cycle time	This displays the time that automatic operation is started from when the automatic start button is pressed in the memory (tape) mode or MDI to when feed hold stop or block stop is applied or the reset button is pressed.

(Note) When the #3 Power ON to #8 Cycle time displays reach the maximum value (9999:59:59), the count stops, and the maximum value remains displayed.

Menus

Menu	Details	Type	Reference
Time setting	This sets the integrated time.	A	2.16.1 Setting the Integrated Time
Time select1	The time that is displayed in upper line on cycle time display area is selected.	C	2.16.2 Setting the Time Display Selection
Time select2	The time that is displayed in bottom line on cycle time display area is selected.	C	
Close	This closes the pop-up window and quits this function	C	

2.16.1 Setting the Integrated Time

Operation method

- (1) Press the menu . The time setting mode is entered. The cursor appears at the "#1 Date" position in the integrated time display.
- (2) Input today's date.
 (Example) 2003/8/19 "2003.08.19" appears at "#1 Date", and the cursor moves to "#2 Time".
- (3) Set the data for each item, and press the key. When "#7 External integrated time 2" has been set, the cursor disappears, and the time setting menu highlight is turned OFF.
 If the item does not need to be set, press the key and move the cursor.
 Note that #8 Cycle time cannot be set, so the cursor will not move.

(Note 1) The data delimiters are as shown below.

Item	Valid delimiters during setting	Delimiters in display
#1 Date	". " or " / "	". "
#2 Time to #7 Ext time2	". : " or " / "	". : "
#8 Cycle time		". : "

(Note 2) If the or key is pressed again during the time setting mode, the time setting mode will be canceled.

Setting range

Display item	Range
#1 Date	1980.1.1 to 2069.12.31
2 Time	00:00:00 to 23:59:59
3 Power ON	00:00:00 to 59999:59:59
4 Auto oper	
5 Auto strt	
6 Ext time 1	
7 Ext time 2	

2.16.2 Setting the Time Display Selection

The displayed time is set on cycle time display area.

Operation method

- (1) Press the menu "Time select1" → The time selection mode is activated, and the following menu appears.

Date	Time	Power ON	Auto oper	Auto strt	Ext time1	Ext time2	Cycle time
------	------	----------	-----------	-----------	-----------	-----------	------------

- (2) Press the time type for the mode to display.
(Example) "Auto oper" → The automatic operation time is displayed in upper line on cycle time display area. The menu display returns to integrated time sub-menu.

- (3) Press the menu "Time select2" → The time selection mode is activated, and the following menu appears.

Date	Time	Power ON	Auto oper	Auto strt	Ext time1	Ext time2	Cycle time
------	------	----------	-----------	-----------	-----------	-----------	------------

- (4) Press the time type for the mode to display.
(Example) "Power ON" → The automatic operation time is displayed in bottom line on cycle time display area. The menu display returns to integrated time sub-menu.

(Note 1) The displayed time is held even after the power is turned OFF and ON.

2.17 Common Variables

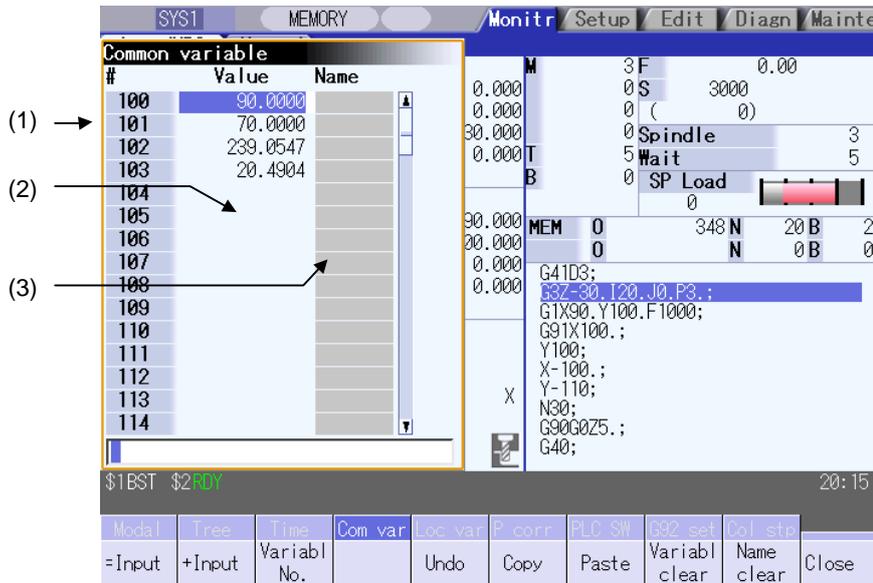


The details of the common variables can be set and displayed on this screen.

If there is a common variable command (Note) in the machining program, the variable value (variable name) set when the block is executed is displayed.

The number of common variable sets differs according to the specifications.

(Note) The variable names can be set for common variables #500 to #519.



Display items

Display item	Details																																				
(1) Variable No.	<p>This is the common variable No. There are the common variable for each part system (#100 to #199) and the common variables shared between the part systems (#400 to #999, #100100 to #800199).</p> <p>The #100 variables appear at the head when the power is turned ON.</p> <p>If "#1052 MemVal" is set to "1" (Designate No. of common variable for common part system), "*" is attached to the part system common variable. (Note that if there is only one part system, "*" will not appear.)</p> <table border="1"> <thead> <tr> <th></th> <th></th> <th>Common variable for each part system</th> <th>Common variables shared between the part systems</th> </tr> </thead> <tbody> <tr> <td rowspan="5">1st part system</td> <td>100 sets</td> <td>500 to 549</td> <td>100 to 149</td> </tr> <tr> <td>200 sets</td> <td>500 to 599</td> <td>100 to 199</td> </tr> <tr> <td>300 sets</td> <td>500 to 699</td> <td>100 to 199</td> </tr> <tr> <td>600 sets</td> <td>500 to 999 100100 to 800199 (Note 7)</td> <td>100 to 199</td> </tr> <tr> <td>700 sets</td> <td>400 to 999 (Note 4) 100100 to 800199 (Note 7)</td> <td>100 to 199</td> </tr> <tr> <td rowspan="5">Multi-part system (n = number of part systems)</td> <td>50 + 50 * n sets</td> <td>500 to 549</td> <td>100 to 149 * n</td> </tr> <tr> <td>100 + 100 * n sets</td> <td>500 to 599</td> <td>100 to 199 * n</td> </tr> <tr> <td>200 + 100 * n sets</td> <td>500 to 699</td> <td>100 to 199 * n</td> </tr> <tr> <td>500 + 100 * n sets</td> <td>500 to 999 100100 to 800199 (Note 7)</td> <td>100 to 199 * n</td> </tr> <tr> <td>600 + 100 * n sets</td> <td>400 to 999 (Note 4) 100100 to 800199 (Note 7)</td> <td>100 to 199 * n</td> </tr> </tbody> </table>			Common variable for each part system	Common variables shared between the part systems	1st part system	100 sets	500 to 549	100 to 149	200 sets	500 to 599	100 to 199	300 sets	500 to 699	100 to 199	600 sets	500 to 999 100100 to 800199 (Note 7)	100 to 199	700 sets	400 to 999 (Note 4) 100100 to 800199 (Note 7)	100 to 199	Multi-part system (n = number of part systems)	50 + 50 * n sets	500 to 549	100 to 149 * n	100 + 100 * n sets	500 to 599	100 to 199 * n	200 + 100 * n sets	500 to 699	100 to 199 * n	500 + 100 * n sets	500 to 999 100100 to 800199 (Note 7)	100 to 199 * n	600 + 100 * n sets	400 to 999 (Note 4) 100100 to 800199 (Note 7)	100 to 199 * n
		Common variable for each part system	Common variables shared between the part systems																																		
1st part system	100 sets	500 to 549	100 to 149																																		
	200 sets	500 to 599	100 to 199																																		
	300 sets	500 to 699	100 to 199																																		
	600 sets	500 to 999 100100 to 800199 (Note 7)	100 to 199																																		
	700 sets	400 to 999 (Note 4) 100100 to 800199 (Note 7)	100 to 199																																		
Multi-part system (n = number of part systems)	50 + 50 * n sets	500 to 549	100 to 149 * n																																		
	100 + 100 * n sets	500 to 599	100 to 199 * n																																		
	200 + 100 * n sets	500 to 699	100 to 199 * n																																		
	500 + 100 * n sets	500 to 999 100100 to 800199 (Note 7)	100 to 199 * n																																		
	600 + 100 * n sets	400 to 999 (Note 4) 100100 to 800199 (Note 7)	100 to 199 * n																																		

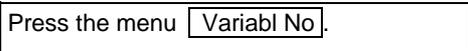
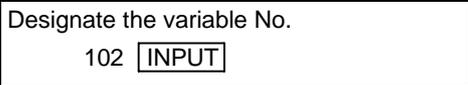
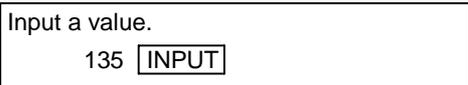
Display item	Details									
	<p>(Note 1) Address #400s common variable can be used only when the sets of common variable is "700 sets" and the parameter "#1336 #400_Valtype" is "1".</p> <p>(Note 2) When the parameter "#1316 CrossCom" is set to "1", the common variables #100100 to #800199 can be used for common variable shared between the part systems. The common variable shared between part systems which can be used is shown in the table below.</p> <table border="1" data-bbox="603 477 1430 808"> <thead> <tr> <th colspan="2" data-bbox="603 477 730 501">Variable sets</th> <th data-bbox="730 477 1430 501">Common variables 1 (When "#1316 CrossCom" = "1")</th> </tr> </thead> <tbody> <tr> <td data-bbox="603 501 730 555">Variable sets</td> <td data-bbox="730 501 874 555">600 sets (500 + 100)</td> <td data-bbox="874 501 1430 555">#100100 to #100199 (Equivalent to # 100 to #199 in 1st part system)</td> </tr> <tr> <td data-bbox="603 555 730 808">specification</td> <td data-bbox="730 555 874 808">700 sets (600 + 100)</td> <td data-bbox="874 555 1430 808">#200100 to #200199 (Equivalent to # 100 to #199 in 2nd part system) #300100 to #300199 (Equivalent to # 100 to #199 in 3rd part system) #400100 to #400199 (Equivalent to # 100 to #199 in 4th part system) #500100 to #500199 #600100 to #600199 #700100 to #700199 #800100 to #800199</td> </tr> </tbody> </table>	Variable sets		Common variables 1 (When "#1316 CrossCom" = "1")	Variable sets	600 sets (500 + 100)	#100100 to #100199 (Equivalent to # 100 to #199 in 1st part system)	specification	700 sets (600 + 100)	#200100 to #200199 (Equivalent to # 100 to #199 in 2nd part system) #300100 to #300199 (Equivalent to # 100 to #199 in 3rd part system) #400100 to #400199 (Equivalent to # 100 to #199 in 4th part system) #500100 to #500199 #600100 to #600199 #700100 to #700199 #800100 to #800199
Variable sets		Common variables 1 (When "#1316 CrossCom" = "1")								
Variable sets	600 sets (500 + 100)	#100100 to #100199 (Equivalent to # 100 to #199 in 1st part system)								
specification	700 sets (600 + 100)	#200100 to #200199 (Equivalent to # 100 to #199 in 2nd part system) #300100 to #300199 (Equivalent to # 100 to #199 in 3rd part system) #400100 to #400199 (Equivalent to # 100 to #199 in 4th part system) #500100 to #500199 #600100 to #600199 #700100 to #700199 #800100 to #800199								
(2) Variable value	<p>The setting range is -999,999,999 to 999,999,999 or blank. An exponential is displayed when a 7 digits are set in the integer section and 5 digits or more as set in the decimal section.</p> <p>(Example) 1234567 → 1.2346E+006, 0.00001 → 1.0000E-005 The minimum setting unit that can be set is 1.0000E-099 (99 digits below decimal point).</p> <p>(Note) When the setting is "blank", the setting will be handled as "0" in the calculations. However, when the conditional expressions EQ or NE are used, the blank will not be handled as "0".</p>									
(3) Variable name	<p>A variable name can be assigned for #500 to #519. Up to seven alphanumeric characters, starting with an alphabet character, can be set for the variable name.</p>									

Menus

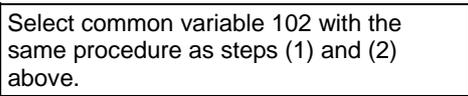
Menu	Details	Type	Reference
=Input	This executes an absolute input.	C	2.17.1 Setting Common Variables
+Input	This executes an addition input.	C	
Variabl No.	This designates the variable No. to be displayed at the head. The common variable for each part system #100 to #199, the common variables shared between the part systems #400 to #999, and the common variables shared between the part systems #100100 to #800199 are displayed in each area. When another area is displayed, press this menu and specify the common variable No.	A	
Undo	This returns the last rewritten data to its original value. This menu key is valid for "Data Input", "Paste" and "Undo" operations. Note that return to the original value is not possible after variable clear and comment clear operations.	C	2.17.2 Copying/Pasting Common Variables
Copy	This copies the variable value and variable name at the cursor position.	A	
Paste	This pastes the copied variable value and variable name.	C	
Variabl clear	This clears the contents of the variable No. at the cursor position or continuous variable Nos. (Note) The contents are not cleared to "0". The state with no data is entered.	A	2.17.3 Erasing Common Variables
Name clear	This clears the variable No. at the cursor position or the continuous variable number's variable name (Variable name: variable No. 500 to 519).	A	
Close	This closes the pop-up window and quits this function.	C	

2.17.1 Setting Common Variables

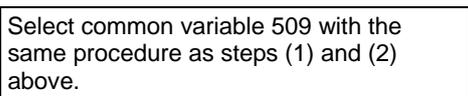
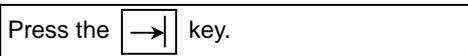
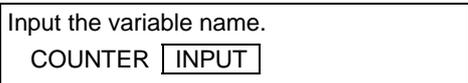
Operation method (Setting "135.000" in variable No. 102)

- (1)  → The menu is highlighted.
- (2)  → The common variable for the set No. appears at the head.
- The cursor can be moved to 102 using the , ,  and  keys.
- (3)  → The set value appears.
- The value can also be set by pressing the menu  instead of the  key.
- The value can also be input as an exponential.
(1.35E2  or 1.35E2 .)

Operation method (Adding "1.234" to variable No. 102)

- (1)  Select common variable 102 with the same procedure as steps (1) and (2) above.
- (2)  → The value obtained by adding the input value to the original value appears.
(Example) If the original value is 135.000, 136.234 appears. ($135.000 + 1.234 = 136.234$)

Operation method (Setting "COUNTER" as the variable name for variable No. 509)

- (1)  Select common variable 509 with the same procedure as steps (1) and (2) above. → The cursor appears at the cursor 509 variable position.
- (2)  Press the  key. → The cursor moves to the 509 variable name.
- (3)  Input the variable name. → "COUNTER" appears.

(Note) The variable name can be set only for variable Nos. 500 to 519.

2.17.2 Copying/Pasting Common Variables

Operation method

- (1) ➔ The cursor moves to the selected variable.
- One of the following methods can be used to select the variable.
- Using the cursor movement keys
 and , move the cursor to the variable to be selected.
 - Press and set the variable No. in the input area, and press the key.
- (2) ➔ One line of the copied variable is highlighted.
- (3) ➔ The copied variable is pasted, and the highlight returns to normal.
The copied variable is held until a variable is newly copied.

2.17.3 Erasing Common Variables

Operation method (Erasing the variable value for variable number 102 to 104)

- (1) Press the menu . ➡ The menu is highlighted.
- (2) Input the No. (range) of the variable to be erased.
 ➡ A message confirming the erasing appears.
 The background color of the erase range (variable value) changes to light blue.
 To erase only one variable, input one variable No.
 (102)
- (3) Press the key. ➡ The selected variable is erased.

Operation method (Erasing the variable name for variable No. 509)

- (1) Press the menu . ➡ The menu is highlighted.
- (2) Select the variable , and press the key
 ➡ A message confirming the erasing appears.
 The background color of the erase range (variable name) changes to light blue.
- (3) Press the key. ➡ The variable name for 509 only is erased.
 The variable value is not erased.

Precautions regarding erasing the variable value and variable name

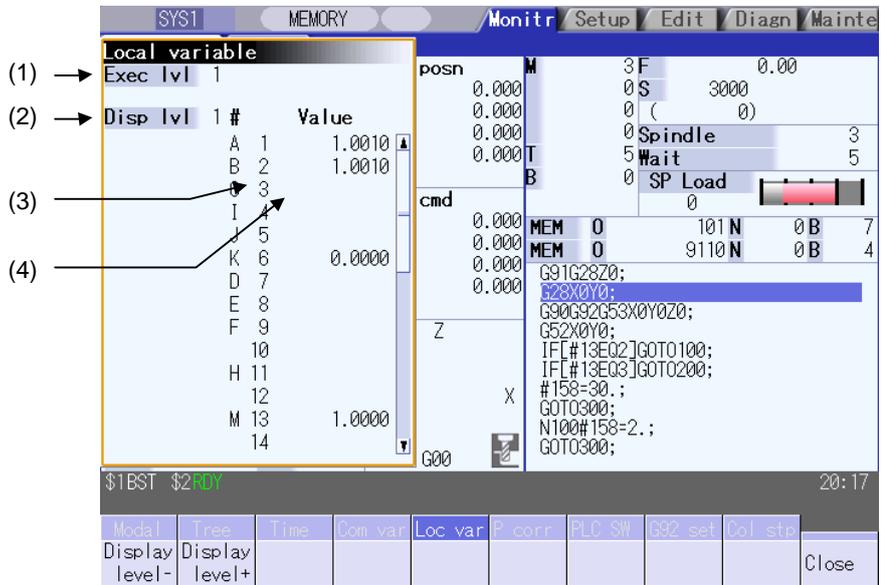
The variable at the cursor position will become the target of the erasing if the key is pressed without designating a variable No.

2.18 Local Variables



The details of the local variables are displayed

Local variables 1 to 33 are prepared for each user macro subprogram call level. Up to 33 local variable data items are displayed on one level. A 5-level configuration from level 0 to level 4 is used in page order. If there is a local variable command or an argument designation called by the user macro subprogram in the block, and that block is executed first, the set variable value (variable name) will be displayed.



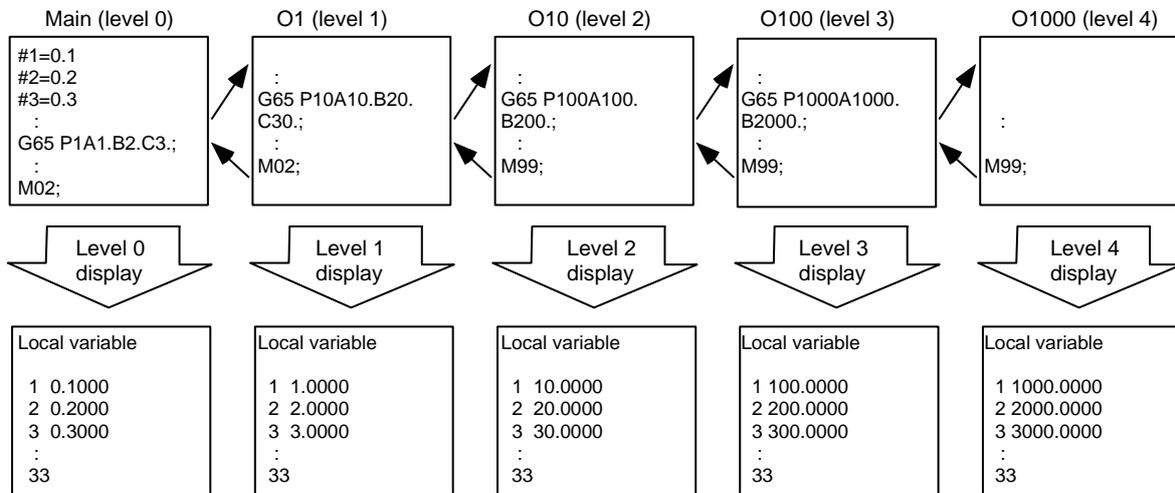
Display items

Display item	Details
(1) Execution level	This displays the nest level of the subprogram control called by the user macro. 0 : Not in user macro call state 1 : User macro call level 1 2 : User macro call level 2 3 : User macro call level 3 4 : User macro call level 4
(2) Display level	This displays the nest level of the local variable displayed in the area.
(3) Variable No.	This displays the local variable No. The alphabetic character before the local variable No. is the argument code. G, L, N, O and P cannot be used as arguments, and thus not displayed. There are 33 local variables (1 to 33) for each user macro subprogram call level.
(4) Variable value	This displays the local variable value. If the variable data is "blank", the display will be blank. An exponential is displayed when a 7 digits are set in the integer section and 5 digits or more as set in the decimal section. (Example) 1234567 → 1.2346E+006, 0.00001 → 1.0000E-005 (Note) When the setting is "blank", the setting will be handled as "0" in the calculations. However, when the conditional expressions EQ or NE are used, the blank will not be handled as "0".

Menus

Menu	Details	Type	Reference
Display level-	This lowers the local variable display level one by one. If this menu is pressed when the display level is 0, the level changes to display level 4.	C	2.18.1 Displaying the Arbitrary Local Variables
Display level+	This increases the local variable display level one by one. If this menu is pressed when the display level is 4, the level changes to display level 0.	C	
Close	This closes the pop-up window and quits this function.	C	

Relation of user macro subprogram call execution level and display level



Precautions

The local variables are not erased when reset or when the power is turned OFF. They are erased at macro call.

2.18.1 Displaying the Arbitrary Local Variables

Operation method (Displaying the next level)

When the current display level is 0

(1) Press the menu .  The level 1 local variables appear from the head.

When the menu is pressed again, the display level changes in the order of 2 → 3 → 4 → 0 → 1 → 2 → ... and so forth.

Operation method (Displaying the previous level)

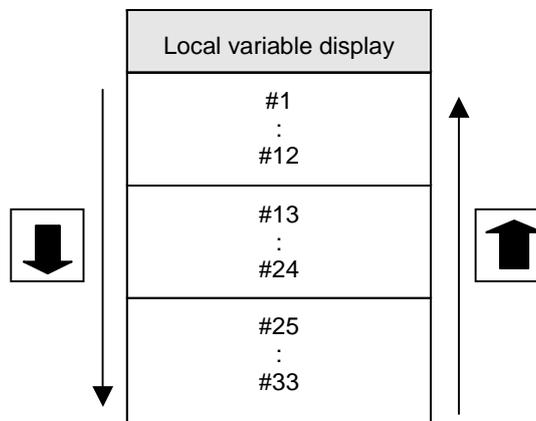
When the current display level is 2

(1) Press the menu .  The level 1 local variables appear from the head.

When the menu is pressed again, the display level changes in the order of 0 → 4 → 3 → 2 → 1 → 0 ... and so forth.

Operation method (Changing the displayed variable No.)

The displayed local variable No. changes when the page changeover keys (previous page , next page ) are pressed.



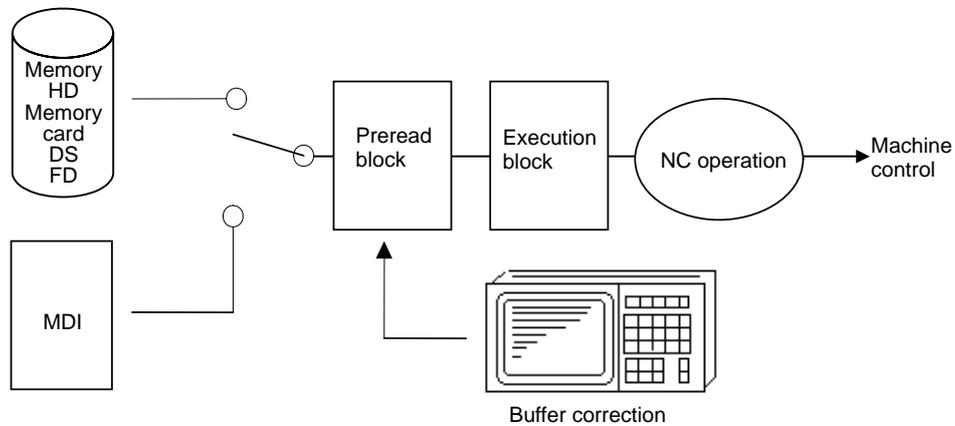
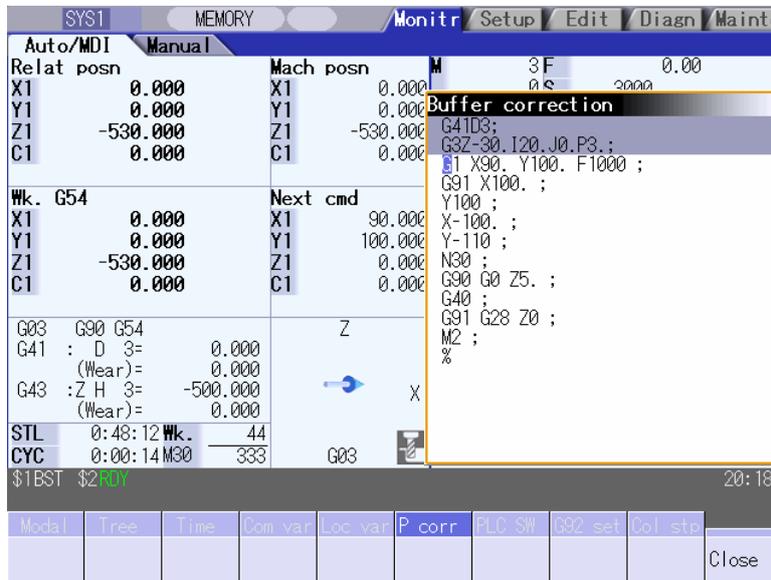
2.19 Buffer Correction



During automatic operation (Memory, HD, FD, DS, memory card) or MDI operation, a block stop can be applied, and the next command can be corrected or changed.

When a program error occurs, the block in which the error occurred can be corrected without resetting the NC, and operation can be continued.

In 70 series, only the memory and the memory card can be used for the input/output device.



- (1) The next command can be corrected in the following two cases.
 - When single block stop is applied, and there is a command block to be corrected in the next command.
 - When there is an error (program error) in the next command and automatic operation is stopped.
- (2) Not only the displayed buffer data but also the contents in the device are corrected with the buffer corrections. (The corrected data is reflected.)
- (3) Several blocks following the next command can be corrected simultaneously.

Menus

Menu	Details	Type	Reference
Close	This closes the pop-up window and quits this function.	C	

Operation method

During a single block stop or when a program error stop occurs, the buffer can be corrected with the following operations, and operation can be continued.

The normally executed program appears in the buffer correction area.

<pre>N121 G28 X10.Y10.; N122 T1212; N123 S1230 M3; N124 G00 X68. Z201.; N125 G01 X80. Z195. F50; N126 Z150.; N127 G02 X100. Z185. R20;</pre>	<p>← Previous command</p> <p>← Command in execution</p> <p>← Next command and subsequent commands</p>
--	---

(1) Press the main menu .



The buffer correction mode is entered.

<pre>N121 G28 X10.Y10.; N122 T1212; N123 S1230 M3; N124 G00 X68. Z201.; N125 G01 X80. Z195. F50; N126 Z150.; N127 G02 X100. Z185. R20;</pre>	<p>← Next command and subsequent commands</p>
--	---

The display changes so that the next command is at the head of the area. The buffer correction area becomes the valid area, and the keys can be input or the cursor moved.

The cursor can be moved freely within the buffer correction area using the cursor keys.

(2) Correct the program with the same method as a normal program editing.

(3) Press the key.



The buffer correction mode ends, and the corrected data is written into program.

If a program error has occurred, the error display is erased.

(4) Confirm that the corrected data is correct, and then restart.



The program execution resumes from the currently stopped position.

Precautions

- (1) Buffer correction mode will not be applied in the following cases.
 - (a) Machining program for buffer corrections is in edit lock or program display lock state.
 - (b) Data protection key 3 is enabled.
 - (c) Machining program for the device in write-protection state
 - (d) Machining program in read-only state
 - (e) During the compound type fixed cycle, the tool center point control or executing the machine tool builder macro program

- (2) Starting and ending buffer corrections
 - (a) If automatic operation is started or there is no next command data, the buffer correction mode is not entered even if the menu key is pressed.
 - (b) To stop or to cancel the buffer corrections, press the menu again, or press the key.
 - (c) If another menu key, etc., is pressed before pressing the key, the corrected buffer contents return to the original data.
 - (d) If changing to another screen or resetting the NC during buffer correction, the buffer correction is canceled. In this case, the corrected contents are not saved, and the buffer correction mode ends.

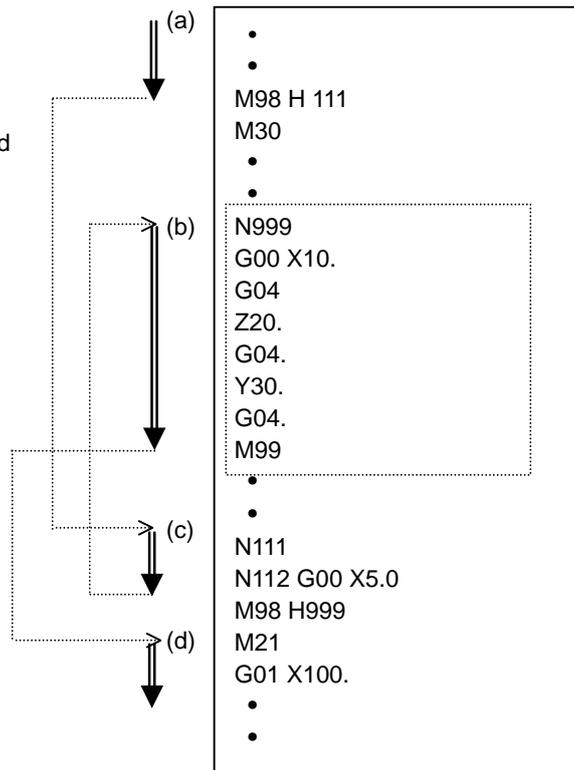
- (3) Display data during buffer correction
 - (a) The key operations used for corrections in the buffer correction mode are the same as the operations when editing a program.
Up to 16 blocks can be displayed. Even when a block is displayed over several lines, all 16 blocks can be displayed by scrolling the buffer correction area.
 - (b) Number of characters in one block is max. 256.
If exceeds 256 characters, the block will be split into two.
 - (c) Up to 4096 characters can be edited with buffer corrections.

- (4) Operations during buffer correction
 - (a) If there is no ";" (EOB) at the data in the last block corrected when the key is pressed, it is added automatically.
 - (b) When the buffer correction mode is entered, the last block displayed in the buffer correction area may not fit in if the block is long, etc.
To correct this type of block, take care to the following points.
 - If a ";" (EOB) is added to the end of the displayed data, the section following the incomplete display becomes another block. If ";" (EOB) is not added, the following section stays the same, and is handled as the continuation of the displayed section.
 - If one block is deleted by pressing the key, the block is deleted up to the section following the incomplete display (up to the ";").
 - (c) Operation cannot be started during buffer correction. An alarm occurs.
 - (d) If NC is reset during buffer correction, buffer correction mode will be canceled, although buffer correction window is left opened. At this time, editing operation is not possible. Close the correction window.
 - (e) Buffer correction writing is not interrupted by NC reset.
 - (f) The message "Can't write into file" appears at any attempt of the buffer mode correction to write into the program being operated or checked, which had stopped when the buffer correction mode started.

- (5) For multi-part system
 - (a) When a machining program under buffer correction is called by another part system, a file input/output error occurs to the caller-side part system, and restart of the operation will be disabled.
 - (b) If the same program is executed in another part system after a buffer correction, there may be some changes in paths, etc. So, please be careful.
 - (c) The program which has searched for the sequence No. while operating in another part system cannot be buffer correction. The message "Buffer correct not possible" appears.

- (6) Previous command and command in execution
 - (a) When the buffer is corrected following the operation mode change at single block stop, nothing will be displayed in previous command and command in execution.
 - (b) When there is no block after sub program call or user macro call, a program error occurs at the last block of the destination program. If a buffer correction is made in this state, the destination program will be displayed in the buffer correction window, but the actual correction result will be reflected on the source program.
 - (c) The buffer cannot be corrected when the block where the buffer is corrected is in the forward side from the block called at another nest level. The message "Buffer correct not possible" appears. The buffer can be corrected when the block where the buffer is corrected is in the rear side from the block called at another nest level.

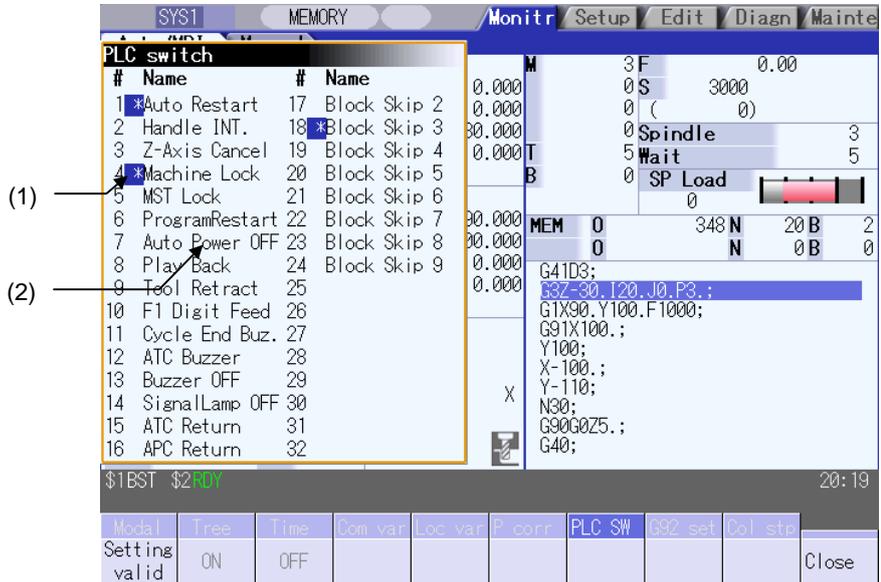
- (a) : The buffer can be corrected.
- (b)(d) : The buffer can be corrected because the block is in the rear side from (a).
- (c) : The buffer cannot be corrected because the block is in the forward side from (b).





2.20 PLC Switch Function

The various control signals for NC operation can be turned ON and OFF. Refer to the instruction manual issued by the machine tool builder for details.



Display items

Display item	Details
(1) Mark indicating switch ON	This is displayed for switches that are turned ON
(2) Switch name	The display contents differ depending on machine tool builder specification.

Menus

Menu	Details	Type	Reference
Setting valid	Setting of the PLC switch is started.	A	2.20.1 Turning PLC Switches ON/OFF
ON	This turns ON the switch currently indicated by the cursor.	C	
OFF	This turns OFF the switch currently indicated by the cursor.	C	
Close	This closes the pop-up window and quits this function.	C	

2.20.1 Turning PLC Switches ON/OFF

Operation method (To turn switch "#6 Program restart" ON)

- (1) Press the menu . ➔ The menu Setting valid will be highlighted.
A message confirming the start of PLC switch setting will appear.
- (2) Press "Y" or ➔ The PLC switch setting mode will be entered.
The cursor will appear at the PLC switch No. position.
The menu keys and will appear normally.
- (3) Press the cursor keys and to move the cursor to "#6". ➔ The #6 number is highlighted.
- (4) Press the menu key . ➔ The "#6 Program restart" switch turns ON, and a mark appears.
To turn OFF, press the menu key OFF.

#	Name
1	*Auto Restart
2	Handle INT.
3	Z-Axis Cancel
4	*Machine Lock
5	MST Lock
6	ProgramRestart
7	Auto Power OFF

#	Name
1	*Auto Restart
2	Handle INT.
3	Z-Axis Cancel
4	*Machine Lock
5	MST Lock
6	*ProgramRestart
7	Auto Power OFF

↖ a mark appears.

(Note 1) If the menu or key is pressed again during the PLC switch setting mode, the PLC switch setting mode will be canceled.

(Note 2) The PLC switch setting mode is canceled when the PLC switch's pop-up window is closed.

(Note 3) The cursor will not appear in modes other than the PLC switch setting mode.

2.21 Verify Stop



The machining program operation can be block stopped at a registered verify stop position. The registered verify stop position can be canceled.

In 70 series, only the memory, the serial, and the memory card can be used for the input/output device.

(Note) Verify stop is an additional specification. The verify stop option is required.

[When verify stop is registered]

The screenshot shows the 'Vrfy stp regist' window with the following data:

File name	Size	Date	Comment
10	701		PARTS1
1001	5252		PARTS2
101	165		PARTS01
103	613		PARTS03
104	591		PARTS04
105	696		PARTS05
109	265		PARTS09
2	428		MOLD01
201	144		MOLD01
210	839		MOLD10
22	428		MOLD22
222	388		MOLD222

At the bottom of the screen, there are buttons for 'Col stp' and 'Close'.

[When verify stop position is displayed]

The screenshot shows the 'Vrfy stp pos' window with the following data:

Device	Memory
O	348
N	20
B	

Below the table, there is a 'Verifying' indicator. Arrows (1), (2), and (3) point to the 'O', 'N', and 'Verifying' elements respectively.

At the bottom of the screen, there are buttons for 'Col stp' and 'Retn'.

Display items

Display items	Details
(1) Verify stop device name	This is the name of the device for which verify stop is to be executed.
(2) Verify stop position	This is the position of the program where verify stop is to be executed. This display is cleared when program stop is completed, or NC is reset or canceled. If the program No. (program name) exceeds 24 characters, "*" will appear as the 24th character.
(3) Display during verification	This is displayed during verification.

Menus (When verify stop is registered)

Menu	Details	Type	Reference
Memory	This selects the device to be stopped for verification. If a device with a directory is selected, the root is selected first. (In 70 series, only "Memory", "Serial", and "Memory card" are displayed.)	C	"Registering the verify stop"
HD		C	
Serial		C	
Memory card		C	
DS		C	
FD		C	
List update	This updates the list contents. (The latest contents of the currently selected device and directory are listed.)	C	
Sort change	This changes the method of sorting the list.	C	2.2.3 Changing the Sorting Method
Stop posn	This opens the verify stop position display window as a pop-up window.	C	"Canceling the verify stop"
Comment nondisp	This changes whether to show or hide the list comment field. The file name field is increased when the comment field is hidden.	B	2.2.2 Changing Whether to Show or Hide the Comment Field
Close	This closes the pop-up window and quits this function.	C	

Menus (When verify stop position is displayed)

Menu	Details	Type	Reference
Stop cancel	This cancels the verify stop setting. Note that automatic operation is not reset.	C	"Canceling the verify stop"
Retn	This returns the list to the <When verify stop is registered>.	C	

Operation method (Registering the verify stop)

- (1) Select the part system for executing verify stop with the key. → The currently selected part system is displayed at the upper left of the screen.
- (2) Press the main menu . → The verify stop submenu appears. The list appears as a pop-up window.
- (3) Select the device.
(Example) → The device name and root directory (HD:/) selected in the device name and directory display fields are displayed.
- (4) Using the , , , and keys, move the cursor to the directory containing the file to be set. → The following is displayed.

```

.          DIR
..         DIR
ABC       DIR
SRAM.BIN  1019904 Mar 06 14:54 2003
    
```

If the list contents differ from the actual device or directory, press the menu .

- (5) Press the key. → The cursor moves within the directory.
- (6) Input the program No., sequence No. and block No. using / as a delimiter.
(Example) 1001/1/2 → The verify stop position is displayed, and the program No. ("MDI" for MDI mode), sequence No., block No. and [Verifying] appear.
- (7) Start the automatic operation. → When the verify stop is completed, a message indicating the end appears. The stop position displayed on the screen is cleared, and the [Verifying] display is erased.

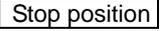
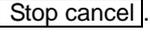
```

.          DIR
..         DIR
123.PRG    62 Dec 20 15:24 2002
68.PRG     62 Dec 20 15:24 2002
69.PRG     166 Dec 20 15:24 2002
    
```

Device	HD
O	1001
N	1
B	2
Verifying	

Device
O
N
B
Verifying

Operation method (Canceling the verify stop)

- (1) Carry out the operations in "Operation method (Registering verify stop)".  The verify stop position appears.
- If the verify stop position is not displayed, press the menu .
- (2) Press the menu .  The stop position displayed on the screen is cleared, and the [Verifying] display is erased.

(Note) Verify stop can also be canceled during operation.

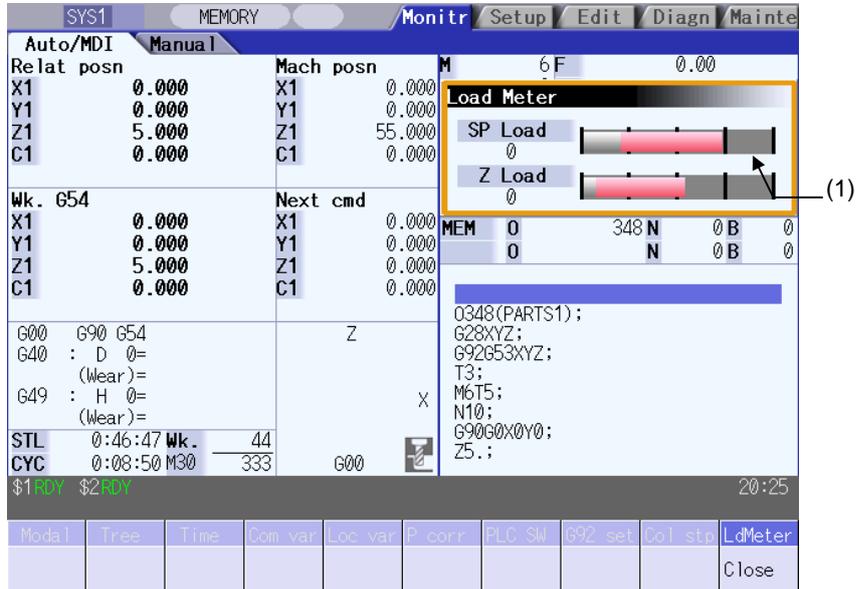
Precautions

- (1) When registering the stop position (O, N, B values), confirm that the block exists in the searched program. Verify stop will not be executed if the corresponding block is not found at the registered stop position.
- (2) Always set one of the O, N or B numbers. If nothing is set, a setting error will occur.
- (3) If the O No. is not set, the O No. used during operation search will be used.
- (4) If the N and B numbers are not set, the block will be searched using the O No.
- (5) If there are several sequences and blocks with the same number in one program, verify stop will be executed after executing the first block that matches in the execution order.
- (6) The settings are canceled when verify stop is executed.
- (7) The settings are canceled when reset is executed.
- (8) If only the program number is set, verify stop will take place at the head of the program only when there is a program number at the first line.
- (9) Verify stop / Verify stop cancel cannot be performed for blocks being executed or blocks already read into the pre-read buffer.
- (10) Verify stop is not performed in the tapping mode.
- (11) Verify stop is possible in a subprogram, but is not possible in a machine tool builder macro program.
- (12) If verify stop is set for a fixed cycle block, verify stop will be executed after the positioning block is completed.
- (13) Verify stop is possible even when editing is locked.
- (14) Verify stop is executed after executing the block set for the verify stop position. If the verify stop position is set in a program call (M98) block, verify stop will be executed before the subprogram is called.
- (15) If the set block is set to be skipped, verify stop will not be executed.
- (16) The verify stop position can be registered for each part system.
- (17) The stop position cannot be registered during verification.
- (18) Registration of verify stop position is not possible for a program with 33 or more file name characters.
- (19) In the case where the device is HD, memory card, DS, or FD at the time of verify stop position registration, inputting "0/" will be resulted in a setting error.

2.22 Load Meter Display



The spindle load and Z axis load, etc., are displayed.



Display items

Display item	Details
(1) Load meter 1 Load meter 2	The spindle load and Z axis load, etc., are displayed in a bar graph. These displayed contents differ according to the machine tool builder.

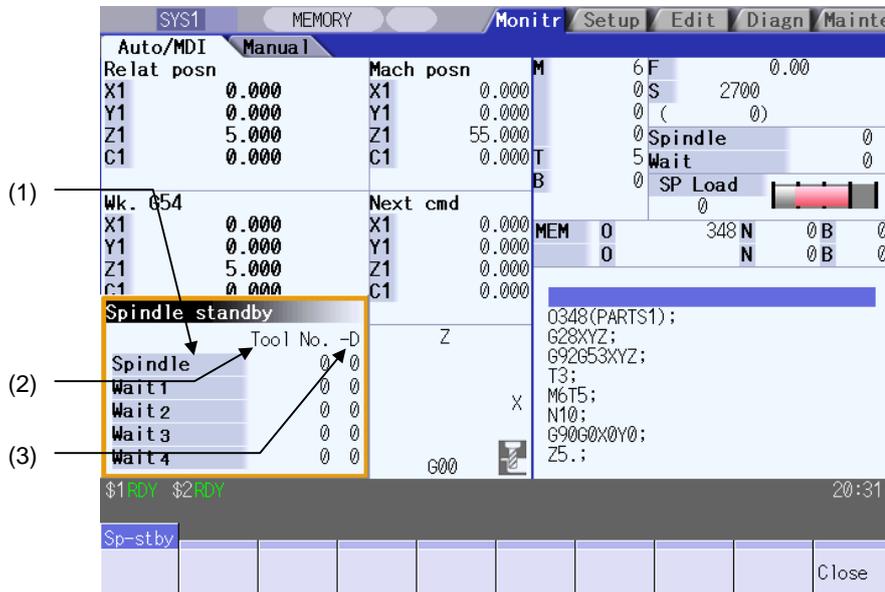
Menus

Menu	Details	Type	Reference
Close	This closes the pop-up window and quits this function.	C	

2.23 Spindle, Standby Display



The current spindle tool No. and the standby tool No. are displayed.



Display items

Display item	Details
(1) Spindle standby	The display contents differ depending on machine tool builder specification.
(2) Tool No.	This displays No. of tool that is attached to the spindle or standby position in the magazine 1
(3) -D	For D data function and purpose, refer to the instruction manual issued by the machine tool builder.

Menus

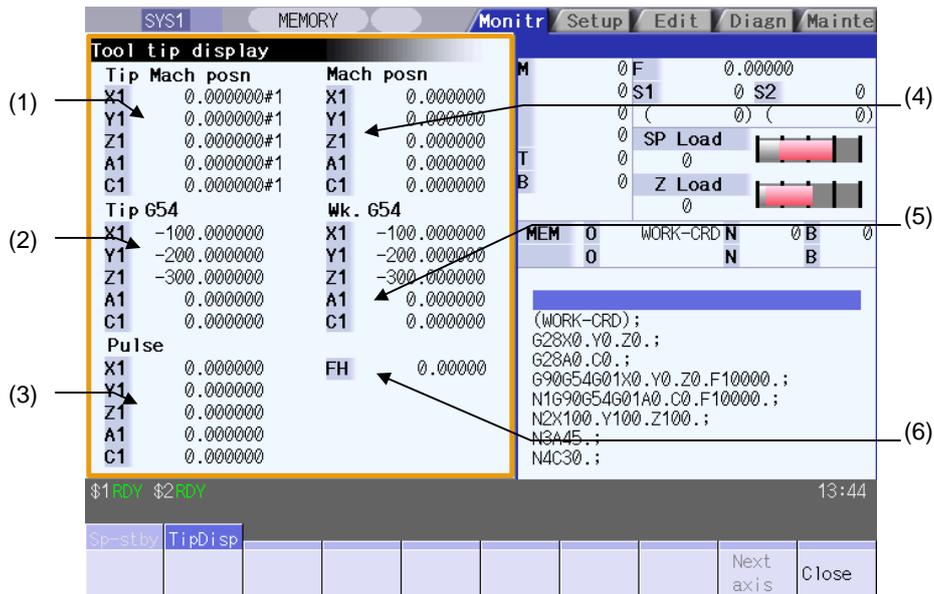
Menu	Details	Type	Reference
Close	This closes the pop-up window and quits this function.	C	



2.24 Tool Center Coordinate Display

The tool center coordinates, handle interrupt amount (tool axis movement) and tool center point speed are displayed during the tool center point control function, tool length compensation along tool axis function, and tool handle feed & interrupt function (handle feed along the tool axis, tool handle interrupt, tool diameter direction handle feed, nose center rotation handle feed).

This function is an additional specification. The 5-axis related options (tool center point control, tool length compensation along the tool axis, tool handle feed & interrupt) are required.



Display items

Display item	Details
(1) Tip Mach posn	This displays the position of the tool center from the machine coordinate system reference position in the machine coordinate system.
(2) Tip G54	This displays the position of the tool center from the workpiece coordinate reference position in the selected workpiece coordinate system.
(3) Pulse	This displays the amount moved in the selected axis direction using the manual pulse generator in the hypothetical machine coordinate system. Basically this is updated only when manual ABS is OFF. If "#7905 NO_ABS" is set to "1", this will be updated regardless of the manual ABS ON/OFF.
(4) Mach posn	This displays the coordinate position of each axis in the basic machine coordinate system having a characteristic position, specified by the machine, as a zero point.
(5) Wk. G54	This displays the selected workpiece coordinate system (G54 to G59)/external workpiece coordinate system (G54.1 Pn) and the workpiece coordinate position in that workpiece coordinate system.
(6) Movement speed of tool center	This displays the movement speed of tool center. By the parameter "#1125 real_f", actual feedrate and command speed can be switched.

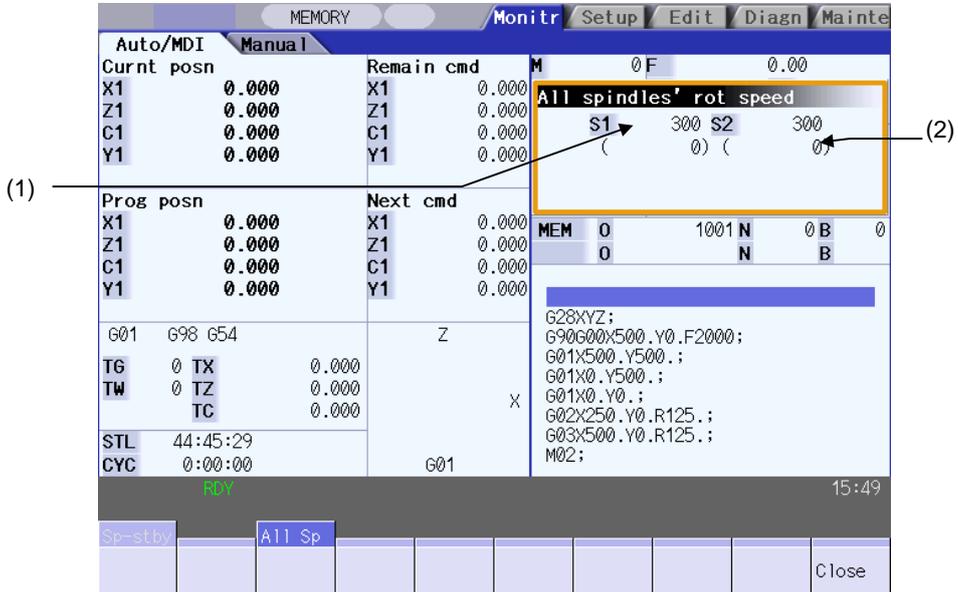
Menus

Menu	Details	Type	Reference
<div data-bbox="196 432 304 488" style="border: 1px solid black; padding: 2px; width: fit-content;">Next axis</div>	<p>This changes the displayed axes each counter. The displayed axes are changed to axes 1 to 5, and axis 6 and over.</p> <p>This menu cannot be selected when applying to the following condition.</p> <ul style="list-style-type: none"> • When the number of enabled axes is 5 or less. • Even if the number of enabled axes is 6 or over, the number of displayed axes is 5 or less according to the parameter "#1069 no_dsp" setting. 	C	
<div data-bbox="196 589 304 645" style="border: 1px solid black; padding: 2px; width: fit-content;">Close</div>	<p>This closes the pop-up window and quits this function.</p>	C	



2.25 All Spindles' Rotation Speed Display

The current spindle tool No. and the standby tool No. are displayed.
 This function only displays the value. It is not possible to operate here.



Display items

Display item	Details
(1) Spindles' command rotation speed	This displays the spindles' command rotation speed (S command value).
(2) Spindles' actual rotation speed	This displays the spindles' actual rotation speed (r/min).

Menus

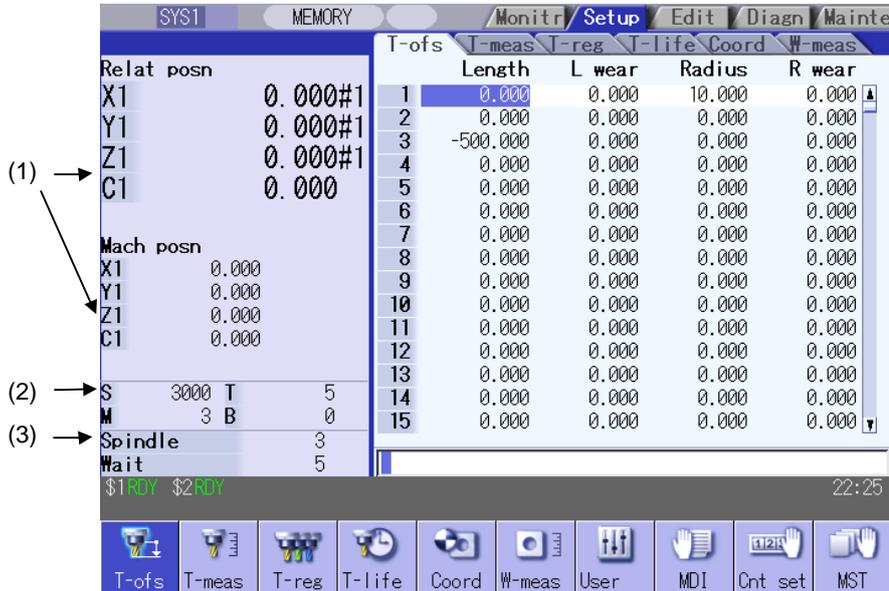
Menu	Details	Type	Reference
Close	This closes the pop-up window and quits this function.	C	

3. Setup Screens

The setup screen is used to make tool and workpiece related settings, set user parameters, edit the MDI, set the counter, and perform manual numerical value commands.

3.1 Screen Configuration

The setup screen is configured as shown below.



Display items

Display item	Details
(1) Counter display	<p>This displays the relative position and machine position counter. Up to 5 axes can be simultaneously displayed.</p> <p>The following status symbols display if each of the axes are in a set position or status.</p> <ul style="list-style-type: none"> #1 to 4 : Reference position 1 to 4 : Servo OFF status MR : Mirror image >< : Axis removed CT : Auxiliary axis state <p>It is possible to set whether to consider the tool length and diameter compensation using parameter "#1287 ext23/bit4".</p> <p>Whether to display the axis counter is set by the parameter "#1069 no_dsp".</p>
(2) M,S,T,B Command	<p>This displays all command values for M (support function command value), S (spindle command rotation speed), T (tool command value) and B (2nd miscellaneous function command value). The spindle command rotation speed displays only 1st spindle.</p> <p>The presence (absence) of the 2nd miscellaneous function command value is designated using parameter "#1170 M2name".</p> <p>Refer to "3.11 Manual Numerical Value Commands" for details.</p>
(3) Spindle/Wait display	<p>The current spindle tool number and standby tool number can be displayed according to the PLC program specifications. (The numbers do not display in the screen if not created with the PLC program.)</p> <p>The contents of this display differ based on the machine tool builder specifications.</p>

Menus

Menu	Details	Reference
 T-ofs	This sets the tool compensation amount. There are three types of tool compensation, and the display differs for each type. The number of tool compensation combinations set/displayed differs depending on the options.	3.2 Tool Compensation amount
 T-meas	This performs the tool measurement. By manually moving the tool to the measurement point, the movement distance from the basic point to the measurement point is measured, and this can be set as the tool compensation amount. This menu displays if the option is enabled.	3.3 Tool Measurement
 T-reg	This registers tools. A tool number is allocated to the tool to allow the NC to recognize the tool attached to the machine. The tool number is registered based on the magazine pot, spindle, and standby location to which the tool is attached.	3.4 Tool Registration
 T-life	This performs the tool life management. The life management data containing information such as the tool usage status is set and displayed. Two management methods are used for tool life management. This menu displays if the option is enabled.	3.5 Tool Life Management
 Coord	This sets the workpiece coordinate system offset amount. This sets and displays the coordinate system offset amount controlled by the NC.	3.6 Workpiece Coordinate System Offset
 W-meas	This performs the workpiece measurement. This menu displays if the option is enabled.	3.7 Workpiece Measurement
 User	This sets the user parameters. This screen is used to switch between, set, and display eleven different parameter types.	3.8 User Parameters
 MDI	This edits the MDI program. Press this menu to show the MDI program contents in a pop-up display.	3.9 MDI Program Editing
 Cnt set	This sets an arbitrary value for the relative position counter.	3.10 Counter Setting
 MST	This performs the manual numerical value commands.	3.11 Manual Numerical Value Commands
 T-list	This searches T code among the designated program (including the subprograms) and lists in order that it was found. This menu is displayed only when the T code list option is valid.	3.12 T Code List
 Pallet	This registers machining programs to the pallet of automatic pallet changer (hereinafter APC). This menu displays if the option is enabled. This menu is only for 700 series.	3.13 Pallet Program Registration

3.2 Tool Compensation Amount



The tool compensation data can be set and displayed.

The tool compensation data screen configuration differs according to the tool compensation type.

The number of tool compensation sets to be set or shown differs according to the option.

Inputting an address key, such as MST, displays manual numerical value command window where manual numerical value command is executed.

(Note) When the tool compensation amount setting during automatic operation is enabled ("#11017 T-ofs set at run" = "1"), the tool compensation amount data can be set even during automatic operation or operation pause.

3.2.1 Tool Compensation Amount (M system)

[Tool compensation type I] : Parameter "#1037 cmdtyp" = 1

The combined amount of the shape compensation and wear compensation are set as the compensation data, with no distinction between shape compensation memory and wear compensation memory. (The tool compensation data is the shape compensation amount + wear compensation amount.)

The compensation data is used commonly for the tool length compensation, tool position offset, tool radius compensation, and three-dimensional tool radius compensation.

Relat posn		T-ofs	T-meas	T-reg	T-life	Coord	#-meas
X1	0.000#1	1	0.000	16	0.000	31	0.000
Y1	0.000#1	2	0.000	17	0.000	32	0.000
Z1	0.000#1	3	-50.000	18	0.000	33	0.000
C1	0.000	4	0.000	19	0.000	34	0.000
		5	0.000	20	0.000	35	0.000
		6	0.000	21	0.000	36	0.000
		7	0.000	22	0.000	37	0.000
		8	0.000	23	0.000	38	0.000
		9	0.000	24	0.000	39	0.000
		10	0.000	25	0.000	40	0.000
		11	0.000	26	0.000	41	0.000
		12	0.000	27	0.000	42	0.000
		13	0.000	28	0.000	43	0.000
		14	0.000	29	0.000	44	0.000
		15	0.000	30	0.000	45	0.000

Mach posn		S	T	M	B	Spindle	Wait
X1	0.000	0	0	0	0	0	0
Y1	0.000	0	0	0	0	0	0
Z1	0.000	0	0	0	0	0	0
C1	0.000	0	0	0	0	0	0

T-ofs	T-meas	T-reg	T-life	Coord	#-meas	User	MCI	Cnt set	MST
=Input	+Input	Wear data	Length data	Nose data	Offset No.	Line copy	Line paste	Line clear	Undo

[Tool compensation type II] : Parameter "#1037 cmdtyp" = 2

The shape compensation amount and wear compensation amount are set separately. The shape compensation amount is furthermore divided into length and radius dimensions.

Of the compensation data, the length dimension data is used for the tool length compensation and tool position offset, and the radius dimension data is used for the tool radius compensation and three-dimensional tool radius compensation.

T-ofs	T-meas	T-reg	T-life	Coord	W-meas
	Length	L wear		Radius	R wear
1	0.000	0.000		10.000	0.000
2	0.000	0.000		0.000	0.000
3	-500.000	0.000		0.000	0.000
4	0.000	0.000		0.000	0.000
5	0.000	0.000		0.000	0.000
6	0.000	0.000		0.000	0.000
7	0.000	0.000		0.000	0.000
8	0.000	0.000		0.000	0.000
9	0.000	0.000		0.000	0.000
10	0.000	0.000		0.000	0.000
11	0.000	0.000		0.000	0.000
12	0.000	0.000		0.000	0.000
13	0.000	0.000		0.000	0.000
14	0.000	0.000		0.000	0.000
15	0.000	0.000		0.000	0.000

Relat posn	X1	0.000#1
	Y1	0.000#1
	Z1	0.000#1
	C1	0.000
Mach posn	X1	0.000
	Y1	0.000
	Z1	0.000
	C1	0.000
S	3000	T 5
M	3	B 0
Spindle		3
Wait		5

CAUTION

- ⚠ If the tool compensation amount or workpiece coordinate system offset amount is changed during automatic operation (including during single block stop), the changes will be valid from the command in the next block or after several subsequent blocks.

Display items

Display item	Details																																																																															
(1) Display area	<p>Displays the tool compensation data. The displayed cursor can be moved and data set. Data that cannot be displayed in the display area can be displayed using the keys below.</p> <p> ,  : Scroll one line at a time.  ,  : Change the display contents one page at a time.</p> <p> Compensation number : This is the compensation data number. Compensation data : The display items differ depending on the tool compensation type. Type I : Compensation data Type II : Length dimension, length wear, radius dimension, radius wear The standard settings and display ranges are as follows.</p> <table border="1" data-bbox="552 734 1449 1122"> <thead> <tr> <th>Type</th> <th>Settings</th> <th>Details</th> <th>Setting/display range</th> </tr> </thead> <tbody> <tr> <td>Type I</td> <td>Compensation data</td> <td>This sets the shape and wear combined compensation amounts.</td> <td>-999.999 to 999.999</td> </tr> <tr> <td rowspan="4">Type II</td> <td>Length dimension</td> <td>This sets the tool length compensation amount.</td> <td rowspan="2">Length/Radius dimension: -999.999 to 999.999</td> </tr> <tr> <td>Length wear</td> <td>This sets the tool length wear compensation amount.</td> </tr> <tr> <td>Radius dimension</td> <td>This sets the tool radius compensation amount.</td> <td rowspan="2">Length/Radius wear: -99.999 to 99.999</td> </tr> <tr> <td>Radius wear</td> <td>This sets the tool radius wear compensation amount.</td> </tr> </tbody> </table> <p>(Note) The display range is when the minimum command unit is 1μm ("#1003 iunit" = B, "#1041 I_inch" = 0).</p> <p>Furthermore, the display and setting ranges are as follows based on the "#1003 iunit" and "#1041 I_inch" setting combination.</p> <table border="1" data-bbox="552 1323 1442 1771"> <thead> <tr> <th rowspan="2">#1003 iunit</th> <th rowspan="2">#1041 I_inch</th> <th colspan="4">Setting/display range</th> </tr> <tr> <th colspan="2">Type II</th> <th colspan="2">Type I</th> </tr> <tr> <td></td> <td></td> <td>Length wear</td> <td>Radius wear</td> <td>Length dimension</td> <td>Radius dimension</td> </tr> </thead> <tbody> <tr> <td rowspan="2">B</td> <td>0</td> <td>-99.999</td> <td>to 99.999</td> <td>-999.999</td> <td>to 999.999</td> </tr> <tr> <td>1</td> <td>- 9.9999</td> <td>to 9.9999</td> <td>-99.9999</td> <td>to 99.9999</td> </tr> <tr> <td rowspan="2">C</td> <td>0</td> <td>-99.9999</td> <td>to 99.9999</td> <td>-999.9999</td> <td>to 999.9999</td> </tr> <tr> <td>1</td> <td>- 9.99999</td> <td>to 9.99999</td> <td>-99.99999</td> <td>to 99.99999</td> </tr> <tr> <td rowspan="2">D</td> <td>0</td> <td>-99.99999</td> <td>to 99.99999</td> <td>-999.99999</td> <td>to 999.99999</td> </tr> <tr> <td>1</td> <td>- 9.999999</td> <td>to 9.999999</td> <td>-99.999999</td> <td>to 99.999999</td> </tr> <tr> <td rowspan="2">E</td> <td>0</td> <td>-99.999999</td> <td>to 99.999999</td> <td>-999.999999</td> <td>to 999.999999</td> </tr> <tr> <td>1</td> <td>- 9.9999999</td> <td>to 9.9999999</td> <td>-99.9999999</td> <td>to 99.9999999</td> </tr> </tbody> </table> <p>(Note) When data without the decimal point is input, a set unit can be specified by the parameter "#8119 Comp. unit switch".</p>	Type	Settings	Details	Setting/display range	Type I	Compensation data	This sets the shape and wear combined compensation amounts.	-999.999 to 999.999	Type II	Length dimension	This sets the tool length compensation amount.	Length/Radius dimension: -999.999 to 999.999	Length wear	This sets the tool length wear compensation amount.	Radius dimension	This sets the tool radius compensation amount.	Length/Radius wear: -99.999 to 99.999	Radius wear	This sets the tool radius wear compensation amount.	#1003 iunit	#1041 I_inch	Setting/display range				Type II		Type I				Length wear	Radius wear	Length dimension	Radius dimension	B	0	-99.999	to 99.999	-999.999	to 999.999	1	- 9.9999	to 9.9999	-99.9999	to 99.9999	C	0	-99.9999	to 99.9999	-999.9999	to 999.9999	1	- 9.99999	to 9.99999	-99.99999	to 99.99999	D	0	-99.99999	to 99.99999	-999.99999	to 999.99999	1	- 9.999999	to 9.999999	-99.999999	to 99.999999	E	0	-99.999999	to 99.999999	-999.999999	to 999.999999	1	- 9.9999999	to 9.9999999	-99.9999999	to 99.9999999
Type	Settings	Details	Setting/display range																																																																													
Type I	Compensation data	This sets the shape and wear combined compensation amounts.	-999.999 to 999.999																																																																													
Type II	Length dimension	This sets the tool length compensation amount.	Length/Radius dimension: -999.999 to 999.999																																																																													
	Length wear	This sets the tool length wear compensation amount.																																																																														
	Radius dimension	This sets the tool radius compensation amount.	Length/Radius wear: -99.999 to 99.999																																																																													
	Radius wear	This sets the tool radius wear compensation amount.																																																																														
#1003 iunit	#1041 I_inch	Setting/display range																																																																														
		Type II		Type I																																																																												
		Length wear	Radius wear	Length dimension	Radius dimension																																																																											
B	0	-99.999	to 99.999	-999.999	to 999.999																																																																											
	1	- 9.9999	to 9.9999	-99.9999	to 99.9999																																																																											
C	0	-99.9999	to 99.9999	-999.9999	to 999.9999																																																																											
	1	- 9.99999	to 9.99999	-99.99999	to 99.99999																																																																											
D	0	-99.99999	to 99.99999	-999.99999	to 999.99999																																																																											
	1	- 9.999999	to 9.999999	-99.999999	to 99.999999																																																																											
E	0	-99.999999	to 99.999999	-999.999999	to 999.999999																																																																											
	1	- 9.9999999	to 9.9999999	-99.9999999	to 99.9999999																																																																											

Menus

Menu	Details	Type	Reference
= Input	This executes an absolute input.	C	3.2.1.1 Setting the Tool Compensation Data
+ Input	This executes an addition input.	C	
Offset No.	Set the compensation number and press <input type="text" value="INPUT"/> to put that number to the top and display the tool compensation data. The cursor moves to the contents of that first line.	A	3.2.1.1 Setting the Tool Compensation Data
Line copy	This copies the contents of the tool compensation data in the selected line (one line).	C	3.2.1.3 Copying/Pasting the Tool Compensation Data
Line paste	The contents (one line) of the copied tool compensation data are written to the line where the cursor is. If changes are made to the data in that line after <input type="text" value="Line copy"/> is selected from the menu, the data prior to the changes is written when pasting. The data in the copied line can be pasted as many times as is required until new data is copied.	C	
Line clear	This erases the compensation data in the selected line (multiple lines can be erased). Selection method: Select the first and the last compensation number to be erased. [Ex.] 1/E: Set all compensation data to "0". If the <input type="text" value="INPUT"/> key is pressed without selecting a line, the compensation data in the line where the cursor is currently positioned is erased.	A	3.2.1.2 Erasing the Tool Compensation Data
Undo	This undoes the last changes to the tool compensation data. (This menu is valid for the "Data Input", "Paste Line" and "Undo" operations.)	C	

3.2.1.1 Setting the Tool Compensation Data

Operation method (Setting "37.000" in the length wear data of compensation No. (102))

(1) Press the menu key

(2) Designate the Compensation No.
102

It is possible to use the , , and keys as well to move the cursor to the offset No.102.

The set number appears at the top of the area, and the cursor moves.

	Length	L wear	Radius	R wear
102	0.000	0.000	0.000	0.000
103	0.000	0.000	0.000	0.000
104	0.000	0.000	0.000	0.000
105	0.000	0.000	0.000	0.000
106	0.000	0.000	0.000	0.000
107	0.000	0.000	0.000	0.000
108	0.000	0.000	0.000	0.000

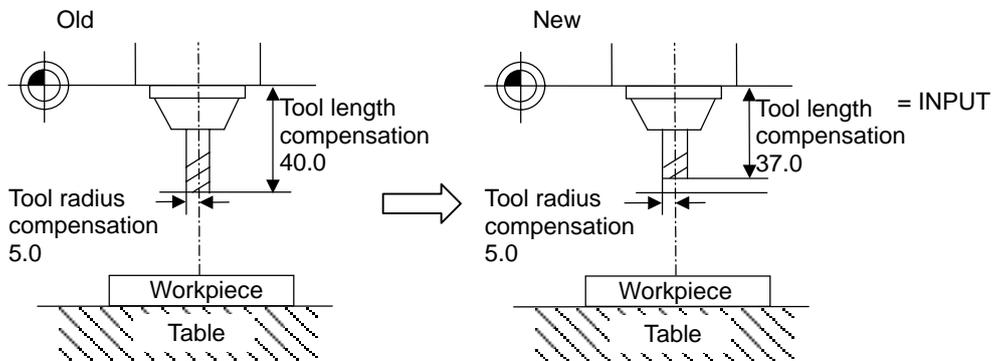
(3) Use the key to move the cursor to the length wear position.

(4) Input the length wear value.
37.000

The set value displays.

Settings can be made by pressing the key instead of the key.

Tool offset data setting example



Operation method (" -3.000" is calculated and set in the length wear data for compensation number (102).)

(1) Use the same procedure as (1) above to move the cursor to the compensation No. 102 length wear position.

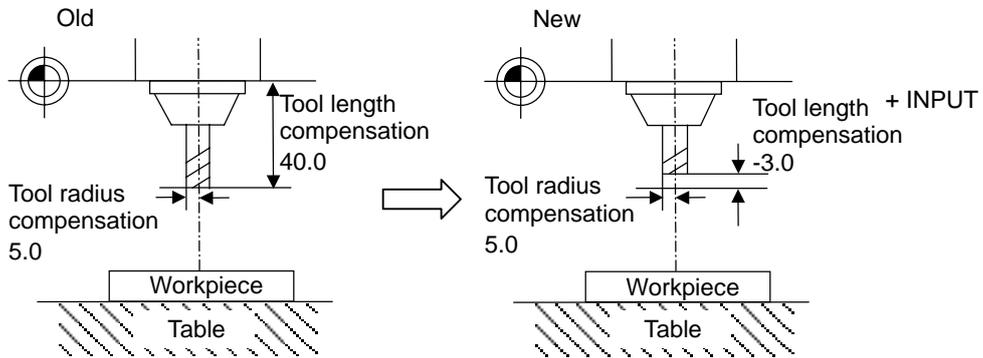
(2) Enter a value.
 -3.000



The input value is added to the original value and displays.

[Ex.] 37.000 displays if the original value is 40.000.
 (40.000 - 3.000 = 37.000)

Tool offset data setting example



3.2.1.2 Erasing the Tool Compensation Data

Operation method (Erasing one line of tool compensation data at the cursor position)

- (1) Move the cursor to the line to be erased with the , , and keys.



The cursor appears at the data item of the selected compensation No.

	Length	L wear	Radius	R wear
102	0.000	0.000	0.000	0.000
103	-280.028	0.000	10.000	0.000
104	-272.083	0.000	0.000	0.000
105	-220.068	0.000	12.000	0.000
106	-290.456	0.000	8.000	0.000
107	-280.001	0.000	0.000	0.000
108	0.000	0.000	0.000	0.000

- (2) Press the and menu.



The menu keys are highlighted and a message displays confirming whether it is okay to erase. The background color of the data to be erased displays in light blue.

- (3) Press the or key.



To cancel the erasing, press a key other than or .

The selected line of data (one line amount) is cleared to zero. The data erased to zero displays at the top of the compensation data.

	Length	L wear	Radius	R wear
102	0.000	0.000	0.000	0.000
103	-280.028	0.000	10.000	0.000
104	-272.083	0.000	0.000	0.000
105	-220.068	0.000	12.000	0.000
106	-290.456	0.000	8.000	0.000
107	-280.001	0.000	0.000	0.000
108	0.000	0.000	0.000	0.000

Operation method (Erasing the compensation data from compensation number 122 to 125.)

- (1) Press the menu.



The menu is highlighted.

- (2) Input the compensation number of the data range to be erased.
122/125



A message box displays to confirm whether it is okay to erase the data. The background color of the data to be erased displays in light blue.

- (3) Press the or key.



Press a key other than or in order not to erase the data.

The compensation data for the selected compensation number is erased, and the menu highlight returns to normal. The data cleared to zero displays at the top of the compensation data.

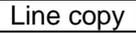
	Length	L wear	Radius	R wear
122	0.000	0.000	0.000	0.000
123	0.000	0.000	0.000	0.000
124	0.000	0.000	0.000	0.000
125	0.000	0.000	0.000	0.000
126	-220.068	0.000	12.000	0.000
127	-290.456	0.000	8.000	0.000
128	-280.001	0.000	0.000	0.000

(Note) If the key is pressed without selecting an compensation number, the line where the cursor is located is erased.

3.2.1.3 Copying/Pasting the Tool Compensation Data

Operation method (Copying/pasting one line of tool compensation data)

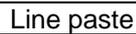
(1) Move the cursor to the line to be copied with the , ,  and  keys

(2) Press the  menu. 

The background color of the copied data changes to light blue.

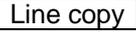
	Length	L wear	Radius	R wear
100	0.000	0.000	0.000	0.000
101	-255.123	0.000	0.000	0.000
102	-250.008	0.000	10.000	0.000
103	-280.028	0.000	10.000	0.000
104	-272.083	0.000	0.000	0.000
105	-220.068	0.000	12.000	0.000
106	-290.456	0.000	8.000	0.000

(3) Move the cursor to the line (compensation No.: 109) where the data is to be pasted.

(4) Press the  menu. 

The copied compensation data is written in the line where the cursor is.
The background color returns to normal.
The copied compensation data is held until another line is copied.

	Length	L wear	Radius	R wear
105	-220.068	0.000	12.000	0.000
106	-290.456	0.000	8.000	0.000
107	-280.001	0.000	0.000	0.000
108	0.000	0.000	0.000	0.000
109	-250.008	0.000	10.000	0.000
110	0.000	0.000	0.000	0.000
111	0.000	0.000	0.000	0.000

(Note 1) If the compensation data for the copied line is changed after the menu  is pressed, the data prior to the change is written when pasted.

3.2.2 Tool Compensation Amount (L system)

3.2.2.1 Wear Data

Set the tool nose wear amount for each tool used. When the tool compensation No. is designated by the tool command (T command), compensation is carried out matching the tool length data and tool nose data.

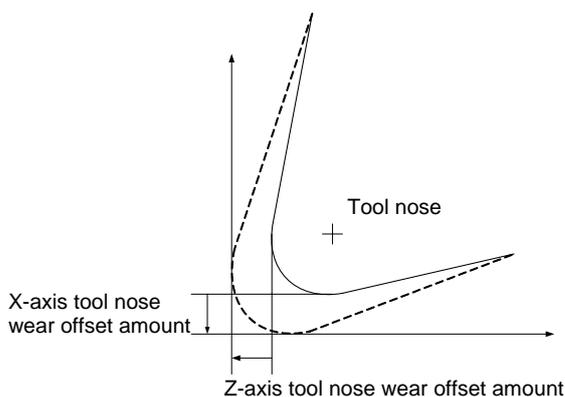
		T-ofs	T-reg	T-life	Coord	X	Y	Z
Relat posn		1				0.000	0.000	0.000
X1	0.000#1	2				0.000	0.000	0.000
Y1	0.000#1	3				0.000	0.000	0.000
Z1	0.000#1	4				0.000	0.000	0.000
C1	0.000	5				0.000	0.000	0.000
		6				0.000	0.000	0.000
		7				0.000	0.000	0.000
Mach posn		8				0.000	0.000	0.000
X1	0.000	9				0.000	0.000	0.000
Y1	0.000	10				0.000	0.000	0.000
Z1	0.000	11				0.000	0.000	0.000
C1	0.000	12				0.000	0.000	0.000
		13				0.000	0.000	0.000
S	0 T 0	14				0.000	0.000	0.000
M	6 B 0	15				0.000	0.000	0.000
Spindle	3							
Wait	5							

CAUTION

If the tool compensation amount or workpiece coordinate system offset amount is changed during automatic operation (including during single block stop), the changes will be valid from the command in the next block or after several subsequent blocks.

Display items

Display item	Details																							
(1) Display area	<p>Displays the tool compensation data. The displayed cursor can be moved and data set. Data that cannot be displayed in the display area can be displayed using the keys below.</p> <p>  ,  : Scroll one line at a time.  ,  : Change the display contents one page at a time. </p> <p>Compensation number : This is the compensation data number. Compensation data : This displays the tool nose wear compensation amount of 1st axis, 2nd axis and additional axis. The standard settings and display ranges are -99.999 to 99.999.</p> <p>(Note 1) The display range is when the minimum command unit is 1μm ("#1003 iunit" = B, "#1041 I_inch" = 0). (Note 2) Set whether to set the additional axis tool nose wear compensation amount for 3rd or 4th axis at parameter "#1520 Tchg34". (Note 3) Displays only for the number of valid axes if only 1st axis or 2nd axis are valid. (Note 4) Depending on the setting of parameter #8010(Max. value), the setting/display range may be different from the above. (Note 5) Depending on the setting of parameter #8011(Max. additional value), the setting/display range may be different from the above.</p> <p>Furthermore, the display and setting ranges are as follows based on the "#1003 iunit" and "#1041 I_inch" setting combination.</p> <p>(Type I, Type II, Type III wear and tool length data)</p> <table border="1"> <thead> <tr> <th>#1003 iunit</th> <th>#1041 I_inch</th> <th>Setting/display range</th> </tr> </thead> <tbody> <tr> <td rowspan="2">B</td> <td>0</td> <td>-99.999 to 99.999</td> </tr> <tr> <td>1</td> <td>- 9.9999 to 9.9999</td> </tr> <tr> <td rowspan="2">C</td> <td>0</td> <td>-99.9999 to 99.9999</td> </tr> <tr> <td>1</td> <td>- 9.99999 to 9.99999</td> </tr> <tr> <td rowspan="2">D</td> <td>0</td> <td>-99.99999 to 99.99999</td> </tr> <tr> <td>1</td> <td>- 9.999999 to 9.999999</td> </tr> <tr> <td rowspan="2">E</td> <td>0</td> <td>-99.999999 to 99.999999</td> </tr> <tr> <td>1</td> <td>- 9.9999999 to 9.9999999</td> </tr> </tbody> </table> <p>(Note) When data without the decimal point is input, a set unit can be specified by the parameter "#8119 Comp. unit switch".</p>	#1003 iunit	#1041 I_inch	Setting/display range	B	0	-99.999 to 99.999	1	- 9.9999 to 9.9999	C	0	-99.9999 to 99.9999	1	- 9.99999 to 9.99999	D	0	-99.99999 to 99.99999	1	- 9.999999 to 9.999999	E	0	-99.999999 to 99.999999	1	- 9.9999999 to 9.9999999
#1003 iunit	#1041 I_inch	Setting/display range																						
B	0	-99.999 to 99.999																						
	1	- 9.9999 to 9.9999																						
C	0	-99.9999 to 99.9999																						
	1	- 9.99999 to 9.99999																						
D	0	-99.99999 to 99.99999																						
	1	- 9.999999 to 9.999999																						
E	0	-99.999999 to 99.999999																						
	1	- 9.9999999 to 9.9999999																						



- X axis offset
X axis tool length offset +X axis wear offset
- Z axis offset
Z axis tool length offset +Z axis wear offset
- Y axis offset (additional axis)
Y axis tool length offset +Y axis wear offset

Menus

Menu	Details	Type	Reference
= Input	This executes an absolute input.	C	3.2.2.4 Operation Method (1) Setting the Tool Compensation Data
+ Input	This executes an addition input.	C	
Wear data	This changes to the wear data display. (Note) This menu displays for Type III.	B	
Length data	This changes to the tool length data display. (Note) This menu displays for Type III.	B	
Nose data	This changes to the tool nose data display. (Note) This menu displays for Type III.	B	
Offset No.	Set the compensation number and press <input type="text" value="INPUT"/> to put that number to the top and display the tool compensation data. The cursor moves to the contents of that first line.	A	3.2.2.4 Operation Method (1) Setting the Tool Compensation Data
Line copy	This copies the contents of the tool compensation data in the selected line (one line).	C	3.2.2.4 Operation Method (3) Copying/Pasting the Tool Compensation Data
Line paste	The contents (one line) of the copied tool compensation data are written to the line where the cursor is. If changes are made to the data in that line after <input type="text" value="Line copy"/> is selected from the menu, the data prior to the changes is written when pasting. The data in the copied line can be pasted as many times as is required until new data is copied.	C	
Line clear	This erases the compensation data in the selected line (multiple lines can be erased). Selection method: Select the first and the last compensation number to be erased. [Ex.] 1/E: Set all compensation data to "0". If the <input type="text" value="INPUT"/> key is pressed without selecting a line, the compensation data in the line where the cursor is currently positioned is erased.	A	3.2.2.4 Operation Method (2) Erasing the Tool Compensation Data
Undo	This undoes the last changes to the tool compensation data. (This menu is valid for the "Data Input", "Paste Line" and "Undo" operations.)	C	

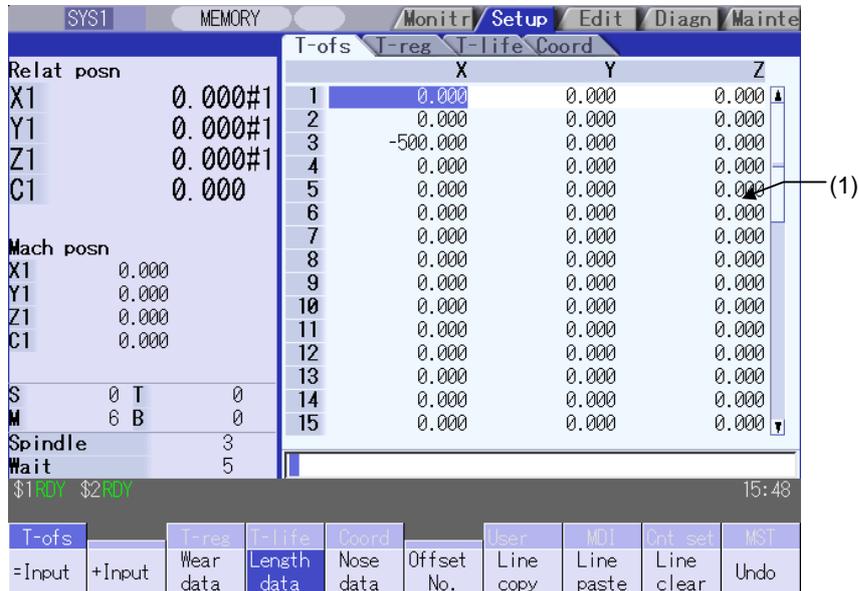
The wear data displays when the power is turned ON.

Following that, the wear data, tool length data, and tool nose data selection is stored.

3.2.2.2 Tool Length Data

Set the tool length in respect to the program basic position of each tool used.

When the tool compensation No. is designated by the tool command (T command), compensation is carried out matching the wear data and the tool nose data.



CAUTION

-  If the tool compensation amount or workpiece coordinate system offset amount is changed during automatic operation (including during single block stop), the changes will be valid from the command in the next block or after several subsequent blocks.

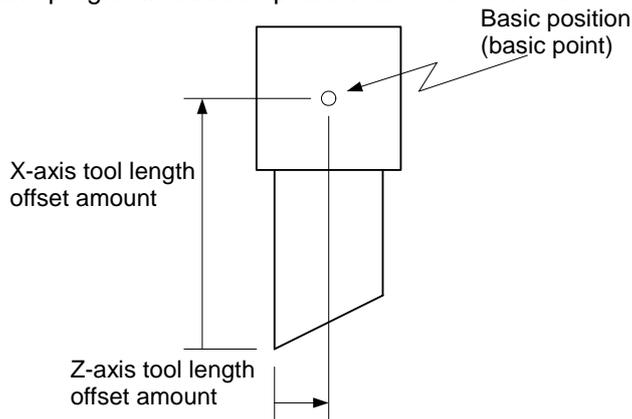
Display items

Display item	Details
(1) Display area (continued on next page)	<p>Displays the tool compensation data. The displayed cursor can be moved and data set. Data that cannot be displayed in the display area can be displayed using the keys below.</p> <p>,  : Scroll one line at a time.</p> <p>,  : Change the display contents one page at a time.</p> <p>Compensation number : This is the compensation data number. Compensation data : This displays the tool length compensation amount of 1st axis, 2nd axis and additional axis.</p> <p>The standard settings and display ranges are -99.999 to 99.999.</p> <p>(Note 1) The display range is when the minimum command unit is 1μm ("#1003 iunit" = B, "#1041 I_inch" = 0). (Note 2) Set whether to set the additional axis tool nose wear compensation amount for 3rd or 4th axis at parameter "#1520 Tchg34". (Note 3) Displays only for the number of valid axes if only 1st axis or 2nd axis are valid.</p>

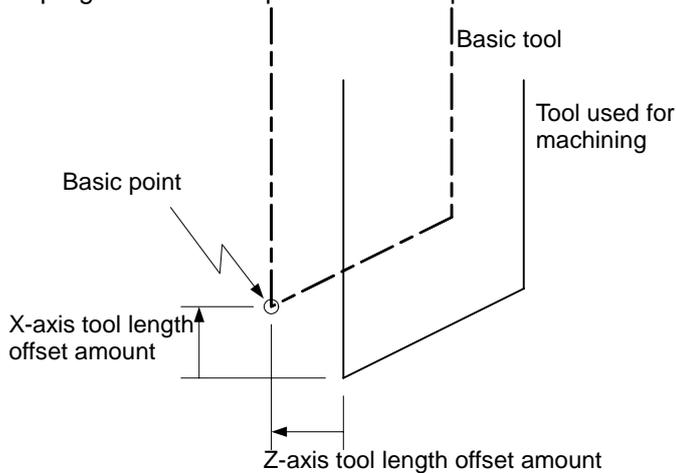
Display item	Details																							
Display area (Continued from previous page)	<p>Furthermore, the display and setting ranges are as follows based on the "#1003 iunit" and "#1041 I_inch" setting combination.</p> <table border="1"> <thead> <tr> <th>#1003 iunit</th> <th>#1041 I_inch</th> <th>Setting/display range</th> </tr> </thead> <tbody> <tr> <td rowspan="2">B</td> <td>0</td> <td>-999.999 to 999.999</td> </tr> <tr> <td>1</td> <td>-99.9999 to 99.9999</td> </tr> <tr> <td rowspan="2">C</td> <td>0</td> <td>-999.9999 to 999.9999</td> </tr> <tr> <td>1</td> <td>-99.99999 to 99.99999</td> </tr> <tr> <td rowspan="2">D</td> <td>0</td> <td>-999.99999 to 999.99999</td> </tr> <tr> <td>1</td> <td>-99.999999 to 99.999999</td> </tr> <tr> <td rowspan="2">E</td> <td>0</td> <td>-999.999999 to 999.999999</td> </tr> <tr> <td>1</td> <td>-99.9999999 to 99.9999999</td> </tr> </tbody> </table> <p>(Note) When data without the decimal point is input, a set unit can be specified by the parameter "#8119 Comp. unit switch".</p>	#1003 iunit	#1041 I_inch	Setting/display range	B	0	-999.999 to 999.999	1	-99.9999 to 99.9999	C	0	-999.9999 to 999.9999	1	-99.99999 to 99.99999	D	0	-999.99999 to 999.99999	1	-99.999999 to 99.999999	E	0	-999.999999 to 999.999999	1	-99.9999999 to 99.9999999
#1003 iunit	#1041 I_inch	Setting/display range																						
B	0	-999.999 to 999.999																						
	1	-99.9999 to 99.9999																						
C	0	-999.9999 to 999.9999																						
	1	-99.99999 to 99.99999																						
D	0	-999.99999 to 999.99999																						
	1	-99.999999 to 99.999999																						
E	0	-999.999999 to 999.999999																						
	1	-99.9999999 to 99.9999999																						

Generally, the center of the tool rest or the tip of the base tool is used as the programmed base position.

(1) The programmed base position is the center of the tool rest:



(2) The programmed base position is the tip of the base tool:



Menus

Same as "3.2.2.1 Wear Data".

3.2.2.3 Tool Nose Data

Set the tool nose radius value (tool nose R), wear radius value (R wear) and tool nose point (tool nose point P) of the tool nose mounted on the tool for each tool used. When the tool compensation No. is designated by the tool command (T command), compensation is carried out matching the tool length data and tool nose data.

Relat posn		T-ofs	T-reg	T-life	Coord
		Nose R	R-wear		Point
X1	0.000#1	1	2.000		1
Y1	0.000#1	2	1.000		2
Z1	0.000#1	3	0.000		0
C1	0.000	4	0.000		0
		5	0.000		0
		6	0.000		0
		7	0.000		0
		8	0.000		0
		9	0.000		0
		10	0.000		0
		11	0.000		0
		12	0.000		0
		13	0.000		0
		14	0.000		0
		15	0.000		0

Mach posn		S	T	M	B	Spindle	Wait
X1	0.000	0	0	6	0	3	5
Y1	0.000						
Z1	0.000						
C1	0.000						

T-ofs	T-reg	T-life	Coord	User	MDI	Ont set	MST
=Input	+Input	Wear data	Length data	Nose data	Offset No.	Line copy	Line paste
						Line clear	Undo

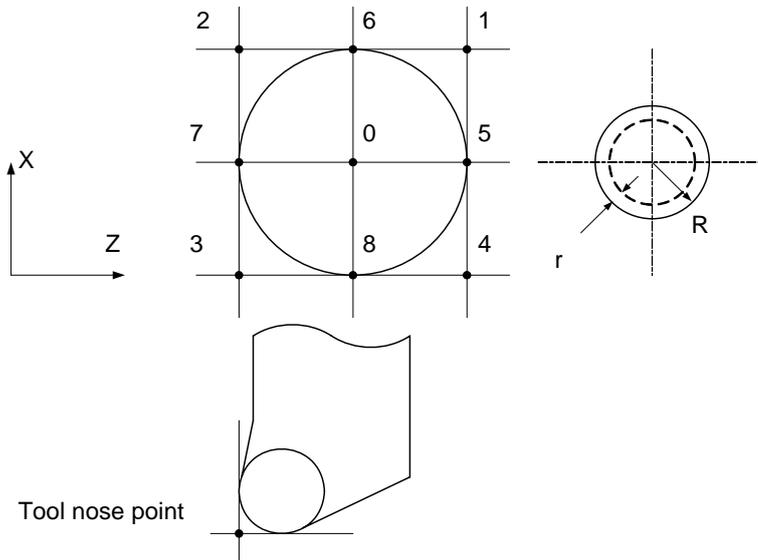
CAUTION

- If the tool compensation amount or workpiece coordinate system offset amount is changed during automatic operation (including during single block stop), the changes will be valid from the command in the next block or after several subsequent blocks.

Display items

Display item	Details												
<p>(1) Display area (continued on next page)</p>	<p>Displays the tool compensation data. The displayed cursor can be moved and data set. Data that cannot be displayed in the display area can be displayed using the keys below.</p> <p>  ,  : Scroll one line at a time.  ,  : Change the display contents one page at a time. </p> <p>Compensation number : This is the compensation data number. Compensation data : This displays the following items: Tool nose R, R wear, tool nose point R</p> <p>The standard settings and display ranges are as follows.</p> <table border="1" data-bbox="552 680 1449 936"> <thead> <tr> <th>Settings</th> <th>Details</th> <th>Setting/display range</th> </tr> </thead> <tbody> <tr> <td>Tool nose R (Note 2)</td> <td>This sets the tool radius (nose R).</td> <td>-999.999 to 999.999</td> </tr> <tr> <td>R wear (Note 2) (Note 3)</td> <td>This sets the tool radius (nose R) wear amount.</td> <td>-99.999 to 99.999</td> </tr> <tr> <td>Tool nose point P (Note 4)</td> <td>This sets the tool nose point (number).</td> <td>0 to 9</td> </tr> </tbody> </table> <p> (Note 1) The display range is when the minimum command unit is 1μm ("#1003 iunit" = B, "#1041 I_inch" = 0). (Note 2) Set at the radius value if parameter "#1019 dia" is "0", or at the diameter value if this parameter value is "1". (Note 3) Depending on the setting of parameter #8010(Max. value), the setting/display range may be different from the above. (Note 4) An absolute setting is applied even if the tool nose point is entered by pressing the  menu and entering a value. </p>	Settings	Details	Setting/display range	Tool nose R (Note 2)	This sets the tool radius (nose R).	-999.999 to 999.999	R wear (Note 2) (Note 3)	This sets the tool radius (nose R) wear amount.	-99.999 to 99.999	Tool nose point P (Note 4)	This sets the tool nose point (number).	0 to 9
Settings	Details	Setting/display range											
Tool nose R (Note 2)	This sets the tool radius (nose R).	-999.999 to 999.999											
R wear (Note 2) (Note 3)	This sets the tool radius (nose R) wear amount.	-99.999 to 99.999											
Tool nose point P (Note 4)	This sets the tool nose point (number).	0 to 9											

Display item	Details					
Display area (Continued from previous page)	Furthermore, the display and setting ranges are as follows based on the "#1003 iunit" and "#1041 I_inch" setting combination.					
	#1003 iunit	#1041 I_inch	Setting/display range			
			Tool nose R		R wear	
	B	0	-999.999	to 999.999	-99.999	to 99.999
		1	- 99.9999	to 99.9999	- 9.9999	to 9.9999
	C	0	-999.9999	to 999.9999	-99.9999	to 99.9999
		1	- 99.99999	to 99.99999	- 9.99999	to 9.99999
	D	0	-999.99999	to 999.99999	-99.99999	to 99.99999
		1	- 99.999999	to 99.999999	- 9.999999	to 9.999999
	E	0	-999.999999	to 999.999999	-99.999999	to 99.999999
1		- 99.9999999	to 99.9999999	- 9.9999999	to 9.9999999	
(Note) When data without the decimal point is input, a set unit can be specified by the parameter "#8119 Comp. unit switch".						



Menus

Same as "3.2.2.1 Wear Data".

3.2.2.4 Operation Method

(1) Setting the Tool Compensation Data

Operation method (Setting "18.000" in the X-axis tool nose wear compensation amount of compensation No. (1))

(1) Press the menu key

(2) Designate the Compensation No.
1

It is possible to use the , , and keys as well to move the cursor to the offset No.1.

The set number appears at the top of the area, and the cursor moves.

	X	Y	Z
1	0.000	0.000	0.000
2	2.000	0.000	0.000
3	15.000	0.000	0.000
4	18.000	0.000	0.000
5	0.000	0.000	0.000
6	0.000	0.000	0.000

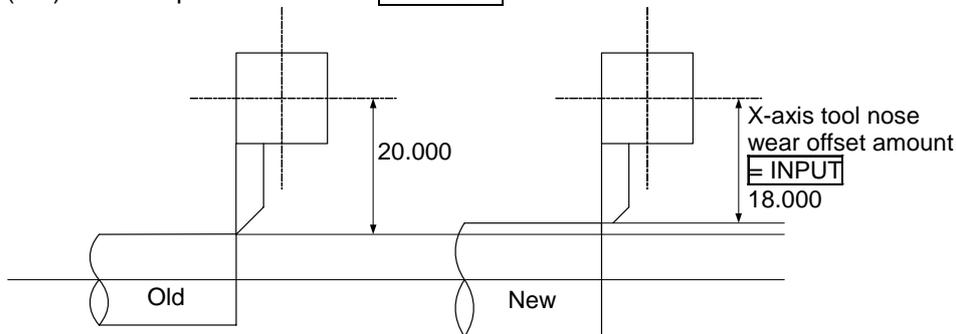
(3) Use the key to move the cursor to the length wear position.

(4) Input the X-axis tool nose wear compensation amount.
18.000

The set value displays.

Settings can be made by pressing the key instead of the key.

(Ex.) Tool compensation data



Operation method (" -2.000" is calculated and set in the X-axis tool nose wear compensation amount for compensation number (102).)

(1) Use the same procedure as (1) above to move the cursor to the compensation No. 1 X-axis length wear position.

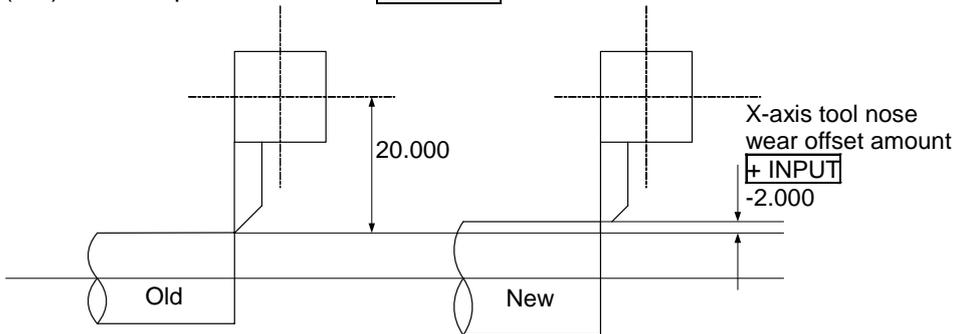
(2) Enter a value.
 -2.000



The input value is added to the original value and displays.

[Ex.] 18.000 displays if the original value is 20.000.
 (20.000 - 2.000 = 18.000)

(Ex.) Tool compensation data



(2) Erasing the Tool Compensation Data

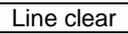
Operation method (Erasing one line of tool compensation data at the cursor position)

- (1) Move the cursor to the line to be erased with the , ,  and  keys.



The cursor appears at the data item of the selected compensation No.

	X	Y	Z
1	16.000	0.000	0.000
2	2.000	0.000	0.000
3	15.000	0.000	0.000
4	18.000	0.000	0.000
5	0.000	0.000	0.000
6	0.000	0.000	0.000

- (2) Press the  and  menu.



The menu keys are highlighted and a message displays confirming whether it is okay to erase. The background color of the data to be erased displays in light blue.

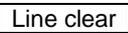
- (3) Press the  or  key.
To cancel the erasing, press a key other than  or .



The selected line of data (one line amount) is cleared to zero. The data erased to zero displays at the top of the compensation data.

	X	Y	Z
2	0.000	0.000	0.000
3	15.000	0.000	0.000
4	18.000	0.000	0.000
5	0.000	0.000	0.000
6	0.000	0.000	0.000
7	0.000	0.000	0.000

Operation method (Erasing the compensation data from compensation number 1 to 3.)

- (1) Press the  menu.



The menu is highlighted.

- (2) Input the compensation number of the data range to be erased.
1/3 



A message box displays to confirm whether it is okay to erase the data. The background color of the data to be erased displays in light blue.

- (3) Press the  or  key.
Press a key other than  or  in order not to erase the data.



The compensation data for the selected compensation number is erased, and the menu highlight returns to normal. The data cleared to zero displays at the top of the compensation data.

	X	Y	Z
1	0.000	0.000	0.000
2	0.000	0.000	0.000
3	0.000	0.000	0.000
4	18.000	0.000	0.000
5	0.000	0.000	0.000
6	0.000	0.000	0.000

(Note) If the  key is pressed without selecting an compensation number, the line where the cursor is located is erased.

(3) Copying/Pasting the Tool Compensation Data

Operation method (Copying/pasting one line of tool compensation data)

(1) Move the cursor to the line to be copied with the , ,  and  keys

(2) Press the menu. 

The background color of the copied data changes to light blue.

	X	Y	Z
1	16.000	0.000	0.000
2	2.000	0.000	0.000
3	15.000	0.000	0.000
4	18.000	0.000	0.000
5	0.000	0.000	0.000
6	0.000	0.000	0.000

(3) Move the cursor to the line (compensation No.: 6) where the data is to be pasted.

(4) Press the menu. 

The copied compensation data is written in the line where the cursor is.
The background color returns to normal.
The copied compensation data is held until another line is copied.

	X	Y	Z
1	16.000	0.000	0.000
2	2.000	0.000	0.000
3	15.000	0.000	0.000
4	18.000	0.000	0.000
5	0.000	0.000	0.000
6	15.000	0.000	0.000

(Note 1) If the compensation data for the copied line is changed after the menu is pressed, the data prior to the change is written when pasted.

(Note 2) It is not possible to paste the data to a different display (wear data, tool length data, tool nose data).



3.3 Tool Measurement

3.3.1 Tool Measurement (M system)

By manually moving the tool to the measurement point, the movement distance from the basic point to the measurement point is measured, and this can be set as the tool compensation amount.

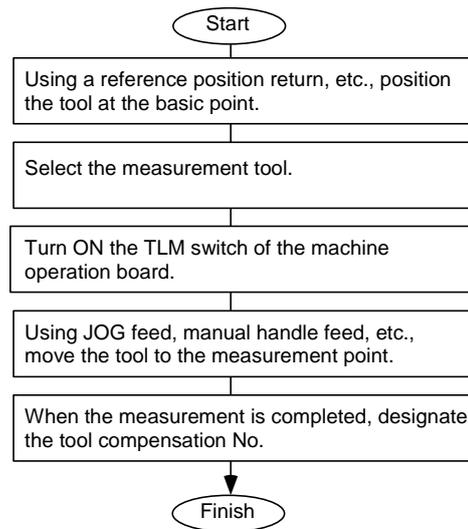
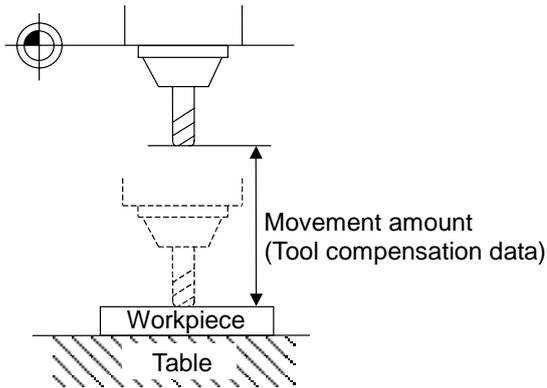
Inputting an address key, such as MST, displays manual numerical value command window where manual numerical value command is executed.

There are two types of tool measurement: Tool length measurement I, Tool length measurement II

■ Tool length measurement I

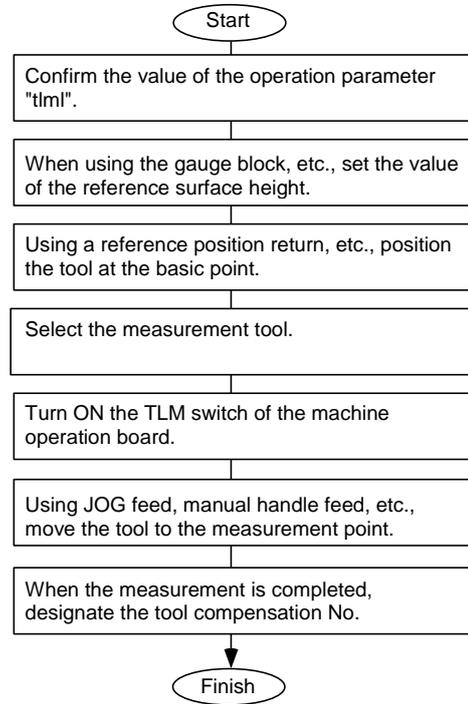
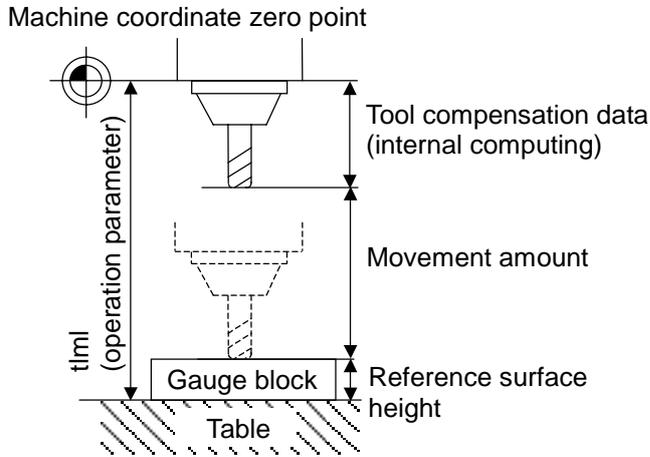
When the tool is at the machine coordinate zero point, the distance from the tool tip to the measurement point (workpiece upper end) is measured, and can be set as tool compensation data.

Machine coordinate zero point



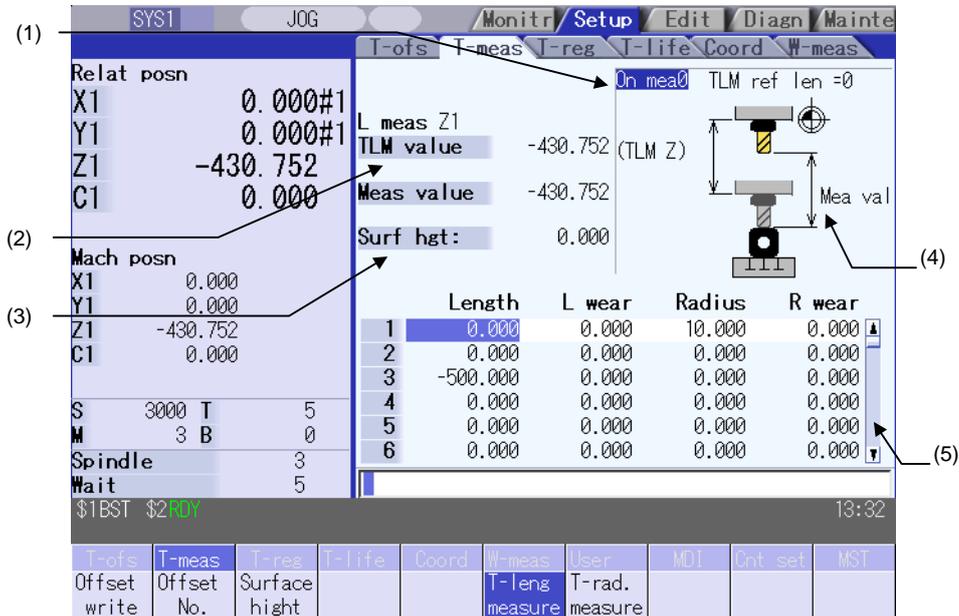
■ Tool length measurement II

When the tool is at the machine coordinate zero point, the distance from the machine coordinate zero point to the tool tip is measured, and can be set as tool compensation data.



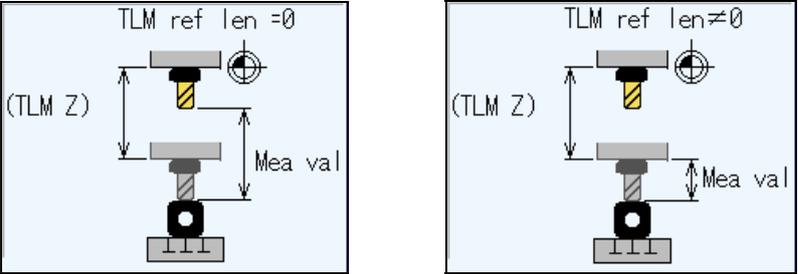
(Note) Changing of the tool measurement type is determined based on the measurement axis (tool length measurement axis, tool radius measurement axis) TLM reference length (parameter "#2016 t_lml+") at each measurement mode. When parameter "#2016 t_lml+" is "0", the tool measurement type is tool length measurement I.

■ Screen image



Display items

Display item	Details
(1) Manual measurement status display	This displays the manual measurement status. Refer to the "Manual measurement status display" section for further details.
(2) Counter display	TLM value : This displays the value during measurement. This will be the same as the machine position until the sensor is contacted. This will be the skip coordinate position after the sensor is contacted. Measurement value : For tool length measurement type I : TLM value – Reference surface height For tool length measurement type II : TLM value – Reference surface height + TLM reference length For tool radius measurement, measurement result is displayed in absolute value. (Note) The TLM value counter differs according to the parameter "#1328 TLM type" value. 0 : The position when the measurement switch was turned ON is displayed as 0. 1 : The display is based on the machine zero point.
L meas : Z (Axis name) R meas : X (Axis name)	The axis selected in the parameters "#8711 TLM L meas axis" and "#8712 TLM D meas axis" is the target axis for the measurement.
(3) Reference surface height	This displays a value for the reference surface height. The setting range is from -99999.999 to 99999.999(mm)

Display item	Details
(4) Guide drawing	<p>This displays the measurement image. The contents of the guide drawing will differ depending on the tool measurement type.</p> <p>Tool length measurement I guide drawing Tool length measurement II guide drawing</p>  <p>The cursor movement, data input and part system changing operation are the same as the tool compensation type (I/II).</p>
(5) Tool compensation data	<p>This displays the tool compensation data. Depending on the tool compensation type (I/II), the display details of the displayed tool compensation data will differ.</p>

Menus

Menu	Details	Type	Reference
Offset write	<p>This writes the value displayed in "Mea value" as the tool compensation amount.</p> <p>The wear amount is cleared to "0" for the tool compensation type II.</p> <p>The tool compensation amount cannot be written in when the menu <input type="text" value="Offset No."/> or <input type="text" value="Surface hight"/> is highlighted.</p> <p>(Note) When the parameter "#8924 MEAS. CONFIRM MSG" is "1", this menu is highlighted and the message "OK? (Y/N)" appears. The measurement result is written as tool compensation amount by pressing <input type="text" value="Y"/> or <input type="text" value="INPUT"/> key. Even if other key is pressed, the result is not written. When this menu is pressed again while displaying the message, the highlight is released and the operation message is erased.</p>	C	
Offset No.	<p>When the compensation No. is set and the <input type="text" value="INPUT"/> key is pressed, the tool compensation data with that number at its head appears. The cursor moves to the length dimension data of the top line of that data.</p>	A	3.2.1 Setting the Tool Compensation Data
Surface hight	<p>This sets the reference surface height data. (The cursor moves to the reference surface height.)</p> <p>When the data is set in the input area and the <input type="text" value="INPUT"/> key is pressed, the reference surface height is set.</p>	A	
T-leng measure	<p>This changes the mode to the tool length measurement mode. This mode is enabled when turning ON the power. (The cursor moves to the length dimension position for tool compensation type II.)</p>	B	Carrying Out Tool Length Measurement
T-rad. measure	<p>This changes the mode to the tool radius measurement mode. (The tool measurement type is valid only for tool length measurement II.) (The cursor moves to the radius dimension position for tool compensation type II.)</p>	B	Carrying Out Tool Radius Measurement

Carrying out tool length measurement

- (1) Turn ON the measurement switch.  The message "On mea 0" appears.
Refer to the PLC Interface Manual for details on measurement switch signals.
- (2) Press the menu.  The menu is highlighted, and the tool length measurement starts. The measurement values are displayed in "Meas value" during the measurement. The menu is highlighted as a default after the power is turned ON.
- (3) Contact the measurement tool against the sensor, using manual feed and manual handle feed. Stop the feed when the tool contacts the sensor.  When the measurement tool contacts the sensor, the skip position displayed in the measurement values counter.
- (4) The cursor moves to the compensation No. that sets the measurement results.  The cursor appears.
 12
The cursor display position differs depending on the measurement mode.
Tool length measurement mode:
Length dimension
Tool radius measurement mode:
Radius dimension
- (5) Press the menu.  The measurement values are written as a tool compensation amount. The wear amount is cleared to "0" for the tool compensation type II.
- (Note 1)** The measurement values are not written if the cursor is in a position other than "Length" or "Radius".
- (Note 2)** The axis returns and stops after contacting the sensor, but be careful not to move the axis after that. If the axis moves after contacting the sensor, the distance that was actually moved will be added to the measurement values counter, and the skip position will not be held.
- (6) Turn OFF the measurement switch to finish measuring.  The message "On mea 0" disappears.

Manual measurement status display

The display will change as shown below during manual measurement.

Display	Meaning
On mea 1	This state is entered if a skip signal is input during manual measurement. The "On mea 2" state is shifted to after deceleration stop is confirmed.
On mea 2	This is the state during the first retract operation. The "On mea 3" state is shifted to after retracting the retract amount.
On mea 3	This is the state in which retract by the retract amount has completed. If the skip after deceleration stop confirmation signal is ON, the status display will remain the same. The "On mea 0" state is shifted to when this state is reset.
On mea 4	This is the state during the second measurement. If the skip signal is not input even after moving to the specified position, a warning will appear, and the status display will remain the same. The "On mea 0" state is shifted to when this state is reset
On mea 5	This state is entered when the skip signal is input during the second measurement. The "On mea 6" state is shifted to after deceleration stop is confirmed.
On mea 6	This is the state during the second return operation. The "On mea 0" state is shifted to after retracting the retract amount.
On mea 0	This state is entered when not in the "On mea 1" to "On mea 6" states.

Carrying out tool radius measurement

The operating procedure of the tool radius measurement and submenu details are the same as when measuring tool length.

(Use instead of)

Refer to "Carrying out tool length measurement" for details.

3.3.2 Tool Measurement (L system)

This is used to carry out manual tool length measurement I or II. Either one is selected depending on the setting of measurement switch and operation mode.

Inputting an address key, such as MST, displays manual numerical value command window where manual numerical value command is executed.

■ Manual tool length measurement I

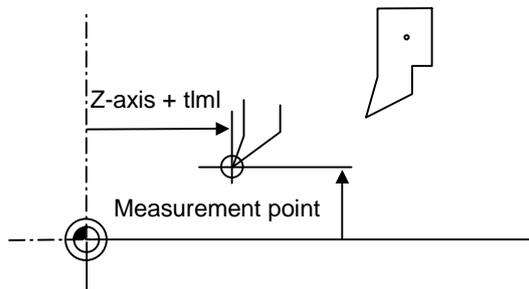
This is the function to calculate the tool length compensation amount automatically by moving the tool to the measurement point with manual feed. There are two types of measurement methods in manual tool length measurement I: the basic point method and the measurement value input method. The method is selected by setting parameter "#1102 tlm".

(1) Basic point method (#1102 tlm = 0)

Obtain the tool length with the tool nose placed on the measurement point.

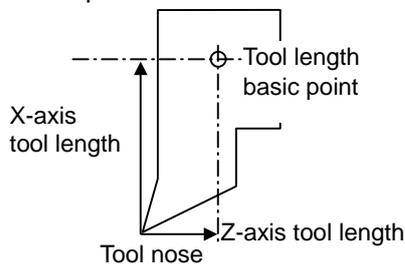
To carry out the basic point method, a point to place the tool nose on (measurement point) is required.

Set the measurement point in parameter "#2015 tlm1-" beforehand.



Tool length = Machine position – Measurement basic point (#2015 tlm1-)

The expression above is used for automatic calculation in the basic point method.



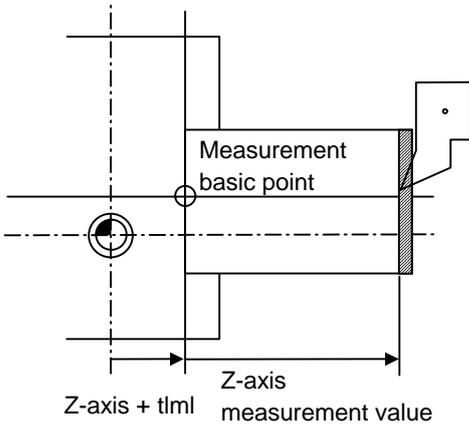
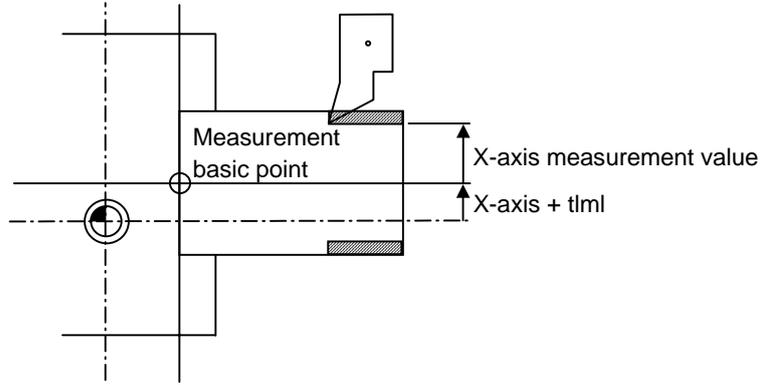
(Note) Always set the measurement point in radius value, regardless of the diameter/radius command.

(2) Measurement value input method (#1102 tlm = 1)

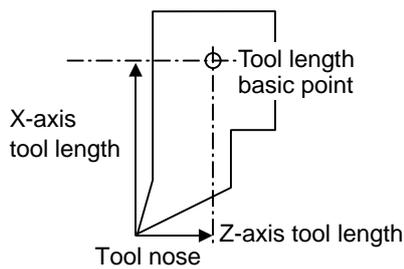
Actually cut the workpiece. Measure its dimensions, and obtain the tool length from the measured values.

The measurement basic point is characteristic for each machine (the center of the chuck face, etc.) To carry out the measurement value input method, a workpiece for measuring is required.

To measure the workpiece, set the basic point (measurement basic point) in parameter "#2015 tlmI-" beforehand.



Tool length = Machine position - Measurement basic point (#2015 tlmI-) - Measurement value
 The expression above is used for automatic calculation in the measurement value input method.

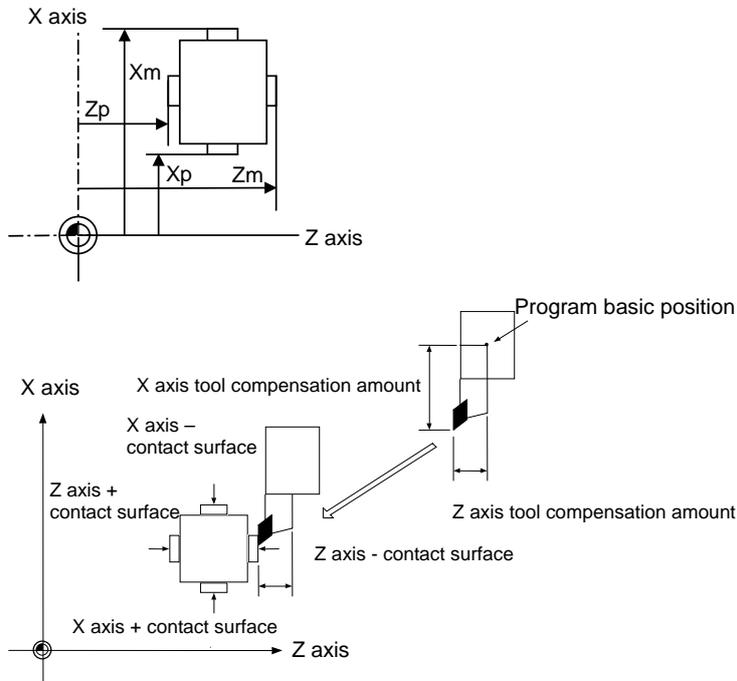


(Note) Always set the measurement basic point in radius value, regardless of the diameter/radius command.

Manual tool length measurement II

By using a device having a touch sensor, the tool compensation amount can be calculated just by contacting the tool nose against the touch sensor with manual feed.

Set the measurement basic position (sensor position) with the parameter "#2015 tml-" and "#2016 tml+" when using the main spindle side, "#2097 tml-" and "#2098 tml+" when using the sub spindle side.



When measured at the main spindle side:

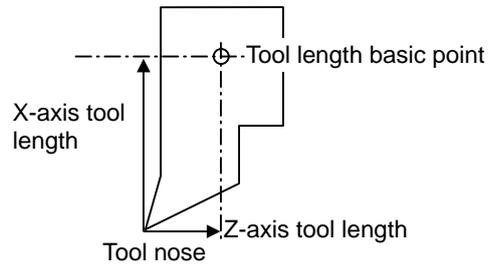
X_m : - sensor machine coordinate position	X axis (position measured by moving in - direction)
→ #2015 tml- X axis	
Z_m : - sensor machine coordinate position	Z axis (position measured by moving in - direction)
→ #2015 tml- Z axis	
X_p : + sensor machine coordinate position	X axis (position measured by moving in + direction)
→ #2016 tml+ X axis	
Z_p : + sensor machine coordinate position	Z axis (position measured by moving in + direction)
→ #2016 tml+ Z axis	

When measured at the sub spindle side:

X_m : - sensor machine coordinate position	X axis (position measured by moving in - direction)
→ #2097 tml2- X axis	
Z_m : - sensor machine coordinate position	Z axis (position measured by moving in - direction)
→ #2097 tml2- Z axis	
X_p : + sensor machine coordinate position	X axis (position measured by moving in + direction)
→ #2098 tml2+ X axis	
Z_p : + sensor machine coordinate position	Z axis (position measured by moving in + direction)
→ #2098 tml2+ Z axis	

Tool compensation amount = Machine coordinate position (machine coordinate position at the time of skip input) - Measurement basic position (sensor position)

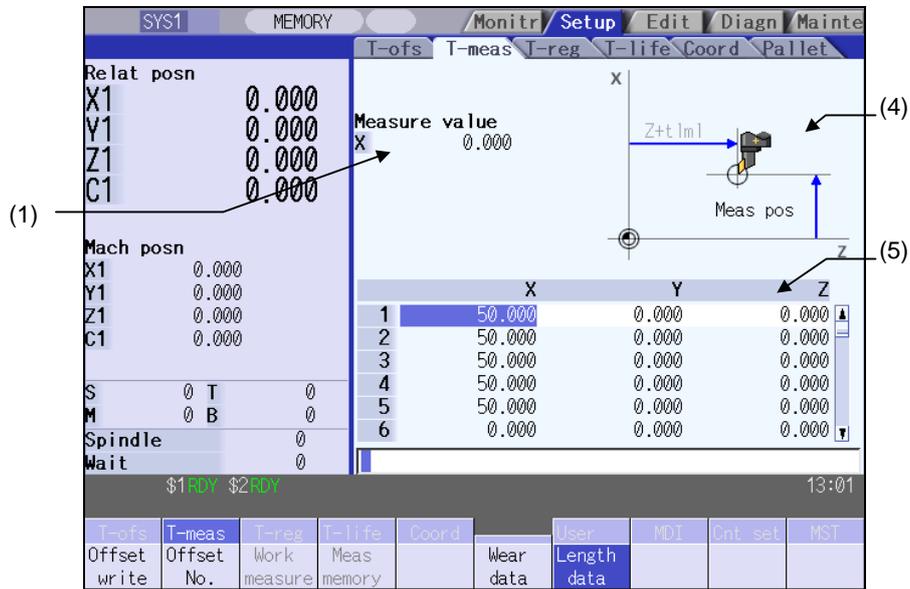
The expression above is used for automatic calculation in Manual tool length measurement II.



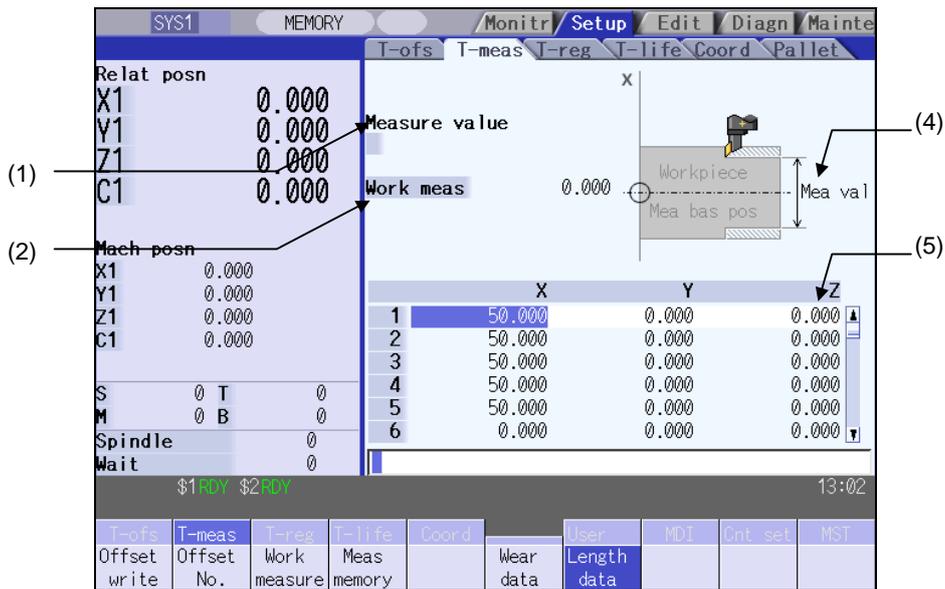
When the tool touches the touch sensor while measurement switch is ON, the calculation result is set in the tool data of the compensation No. specified with "Tool No. (Tool length measurement 2)" (PLC device), and the wear data of the compensation No. set in "Wear compensation No. (Tool presetter)" (PLC device) is cleared.

■ Screen image

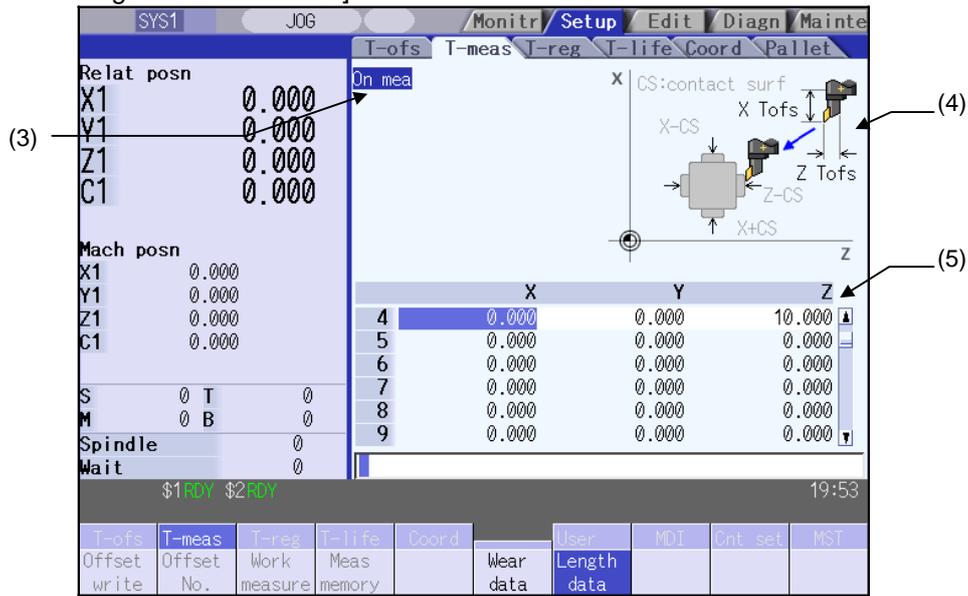
[Manual tool length measurement I: Basic point method]



[Manual tool length measurement I: Measurement value input method]

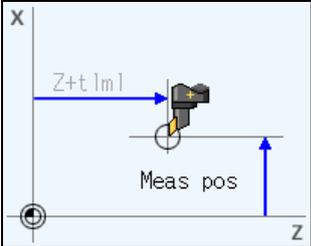
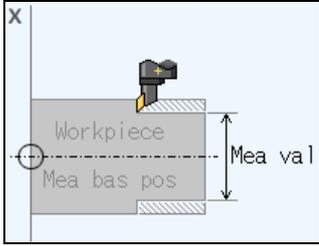
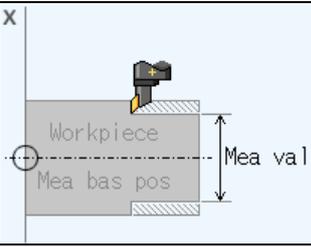


[Manual tool length measurement II]



Display items

Display item	Details
(1) Measure value	Measurement result and measurement axis are displayed. Displayed timing and measurement result differ, depending on basic point method or measurement value input method. <ul style="list-style-type: none"> • Basic point method Measurement result for the axis at the cursor position is displayed at all time. Displayed data: Machine position - Measurement point (#2015 tml-) • Measurement value input method Measurement result is displayed when the Meas memory menu is pressed. Displayed data: Machine position - Measurement basic point (#2015 tml-) (Note) This is displayed only in "Manual tool length measurement I".
(2) Work meas	Workpiece measurement result is set and displayed. Setting is possible by pressing the Work measure menu or touching the value area of the workpiece measurement value. Setting range: -99999.999 to 99999.999(mm) (Note) This is displayed only in "Manual tool length measurement I: Measurement value input method".
(3) On mea	"On mea" is displayed when the measurement switch is ON. (Note) This is displayed only in "Manual tool length measurement II".

Display item	Details
(4) Guide drawing	<p>This displays the measurement image. The contents of the guide drawing will differ depending on the tool measurement type.</p> <p><Manual tool length measurement I: Basic point method></p>  <p><Manual tool length measurement I: Measurement value input method></p>  <p><Manual tool measurement II></p> 
(5) Tool compensation data	<p>This displays the tool compensation data.</p> <p>(Note 1) Cursor movement, data input and part system changing operation are same as "Tool compensation type III (L system)".</p> <p>(Note 2) Automatically jumps to the compensation No. registered in "Tool No. (Tool length measurement 2)" (PLC device) during manual tool length measurement II. (Jumps even when the wear data is being displayed.) Note that the , , ,  keys are disabled.</p>

(Note) Measurement result and workpiece measurement value will be cleared when changing of part systems, screens and measurement modes are carried out.

Menus

Menu	Details	Type	Reference
Offset write	<p>Depending on the mode, the following value is input as the tool compensation amount at the cursor position, and the wear amount is cleared to "0".</p> <ul style="list-style-type: none"> Manual tool length measurement I: Basic point method "Measure value" value Manual tool length measurement I: Measurement value input method "Measure value" value - "Work meas" value <p>In "Manual tool length measurement I: Measurement value input method", "Measure value" and "Work meas" will be cleared after compensation amount is written.</p> <p>(Note 1) Tool compensation amount cannot be written in the following cases:</p> <ul style="list-style-type: none"> When the menu <input type="text" value="Offset No."/> and <input type="text" value="Work measure"/> are highlighted When the wear data is being displayed When the measurement result exceeds the setting range <p>(Note 2) If "Manual tool length measurement II", this is displayed in gray and cannot be selected.</p> <p>(Note 3) When the parameter "#8924 MEAS. CONFIRM MSG" is "1", this menu is highlighted and the message "OK? (Y/N)" appears. The measurement result is written as tool compensation amount by pressing Y or INPUT key. Even if other key is pressed, the result is not written. When this menu is pressed again while displaying the message, the highlight is released and the operation message is erased.</p>	C	<p>Carrying out tool measurement (Manual tool length measurement I: Basic point method)</p> <p>Carrying out tool measurement (Manual tool length measurement I: Measurement value input method)</p>
Offset No.	<p>This sets the compensation No. and press the <input type="text" value="INPUT"/> key. The tool compensation data is displayed, placing the No. at the top.</p> <p>(Note) If the measurement switch is turned ON, this is displayed in gray and cannot be selected.</p>	A	3.2.1 Setting the Tool Compensation Data
Work measure	<p>This sets the workpiece measurement value. If this menu is pressed, the cursor moves to the workpiece measurement value. Point the cursor to the axis to be set, and set the data in the data setting area. Press the <input type="text" value="INPUT"/> key to set the workpiece measurement value.</p> <p>(Note) If not "Manual tool length measurement I: Measurement value input method", this is displayed in gray and cannot be selected.</p>	A	Carrying out tool measurement (Manual tool length measurement I: Measurement value input method)
Meas memory	<p>Measurement result is calculated from the current machine position and displayed.</p> <p>(Note) If not "Manual tool length measurement I: Measurement value input method", this is displayed in gray and cannot be selected.</p>	C	<p>Carrying out tool measurement (Manual tool length measurement I: Basic point method)</p> <p>Carrying out tool measurement (Manual tool length measurement I: Measurement value input method)</p>
Wear data	<p>Wear data is displayed in tool compensation data.</p>	B	
Length data	<p>Tool length data is displayed in tool compensation data. This is the default mode when the power is initially turned ON.</p>	B	

(Note) The data displayed on the tool compensation amount screen and the data displayed on "Tool compensation data" in the tool measurement screen are not linked one another. (The wear data is not necessarily displayed when the tool measurement screen has opened while the wear data is displayed on the tool compensation amount screen.)

Carrying out tool length measurement (Manual tool length measurement I: Basic point method)

(Example) Carrying out tool measurement for compensation No.2, Z axis

(1) Select the tool to be measured with manual numerical value command. (Refer to "3.11 Manual Numerical Value Command" for details.)



The tool to be measured is selected.
(Note) Select a tool without specifying compensation No. If compensation No. is specified, measurement result may not be obtained correctly.

(2) Point the cursor to the position (compensation No. 2, Z axis) where the measurement result are to be set.



The cursor is displayed at compensation No.2, Z axis. Calculation result of Z axis is displayed in "Measure value".

Measure value	
Z	10.000

(Note) In order to measure a different axis, point the cursor to the axis to be measured and perform the following operations.

(3) Press the **Offset write** menu.



The value displayed in "Measure value" is set in compensation No.2, Z axis.

	X	Z	C
1	0.000	0.000	0.000
2	0.000	10.000	0.000
3	0.000	0.000	0.000
4	0.000	0.000	0.000
5	0.000	0.000	0.000
6	0.000	0.000	0.000

(Note 1) Compensation amount is written in diameter value for diameter command, in radius value for radius command.

Carrying out tool measurement (Manual tool length measurement I: Measurement value input method)

(Example) Carrying out tool measurement for compensation No.3, X axis

(1) Select the tool to be measured with manual numerical value command. (Refer to "3.11 Manual Numerical Value Command" for details.)



The tool to be measured is selected.
(Note) Select a tool without specifying compensation No. If compensation No. is specified, measurement result may not be obtained correctly.

(2) Point the cursor to the position (compensation No. 3, X axis) where the measurement result are to be set.



The cursor is displayed at compensation No.3, X axis.
(Note) In order to measure a different axis, point the cursor to the axis to be measured and perform the following operations.

(3) Cutting is executed in the longitudinal direction. (For the Z axis, face turning is executed.)



(4) The **Meas memory** menu is pressed without retracting the tool at the end of cutting operation.



Measurement result is displayed.

Measure value	
X	20.000

- (5) Retract the tool and measure the workpiece. And then press the **Work measure** menu.
- The **Work measure** menu is highlighted and the cursor moves to the workpiece measurement value.
- | | |
|-----------|-------|
| Work meas | 0.000 |
|-----------|-------|
- (6) Set the workpiece measurement result. 10 **INPUT**
- 10 is set for the workpiece measurement value. Highlight of the **Work measure** menu is canceled and the cursor moves to the tool compensation data.
- | | |
|-----------|--------|
| Work meas | 10.000 |
|-----------|--------|
- (Note)** Set a diameter value for diameter command and a radius value for radius command.
- (7) Press the **Offset write** menu.
- "Measurement result - workpiece measurement value" is set to compensation No.3, X axis.
- | | X | Y | Z |
|---|--------|--------|-------|
| 1 | 0.000 | 0.000 | 0.000 |
| 2 | 0.000 | 10.000 | 0.000 |
| 3 | 10.000 | 0.000 | 0.000 |
| 4 | 0.000 | 0.000 | 0.000 |
| 5 | 0.000 | 0.000 | 0.000 |
| 6 | 0.000 | 0.000 | 0.000 |

(Note 1) Compensation amount is written in diameter value for diameter command, in radius value for radius command.

Carrying out tool measurement (Manual tool length measurement II)

- (1) Select the tool to be measured with manual numerical value command. (Refer to "3.11 Manual Numerical Value Command" for details.)
- The tool to be measured is selected.
- (2) Set the compensation No. of the compensation data to be set in "Tool No. (Tool length measurement 2)" (PLC device).
- The compensation No. of the compensation data to be set is selected.
- (3) Set the compensation No. of the wear data to be cleared in "wear compensation No. (tool presetter)" (PLC device).
- The compensation No. of the wear data to be cleared is selected.
- (4) Turn the measurement switch ON.
- "On meas" is displayed. Guide drawing for manual tool length measurement II is displayed. Tool compensation data is displayed, placing the compensation No. set in "Tool No. (Tool length measurement 2)" (PLC device) at the top.
- | | X | Y | Z |
|---|-------|-------|-------|
| 4 | 0.000 | 0.000 | 0.000 |
| 5 | 0.000 | 0.000 | 0.000 |
| 6 | 0.000 | 0.000 | 0.000 |
- (Note)** The row at the cursor position will not be changed.

- (5) Move the additional axis in the minus direction by manual feed and have the tool nose contact with the touch sensor.



Measurement value is set for the additional axis of the compensation No. set in "Tool No. (Tool length measurement2)" (PLC device)

	X	Y	Z
4	0.000	0.000	10.000
5	0.000	0.000	0.000
6	0.000	0.000	0.000

- (6) Turn the measurement switch OFF.



"On meas" disappears.

- (Note 1)** When entering the sensor area, the axis can be moved only in one direction selected from +X, -X, +Z, -Z, (+Y, -Y). If two axes are moved simultaneously, measurement will not be carried out. At this time, "E78 AX UNMATCH (TLM)" is displayed and the axis movement stops.
- (Note 2)** If the tool nose is contacting the sensor, the axis can be moved only in the direction moving away from the sensor. Whether or not the tool nose is detached from the sensor can be judged by the following conditions.
- The sensor signal has been turned OFF for 500ms or more.
 - The tool nose is moved 100 μ m or more after the sensor signal has turned OFF.
- The above conditions are set with parameter "#1227 aux11/bit2".

3.4 Tool Registration

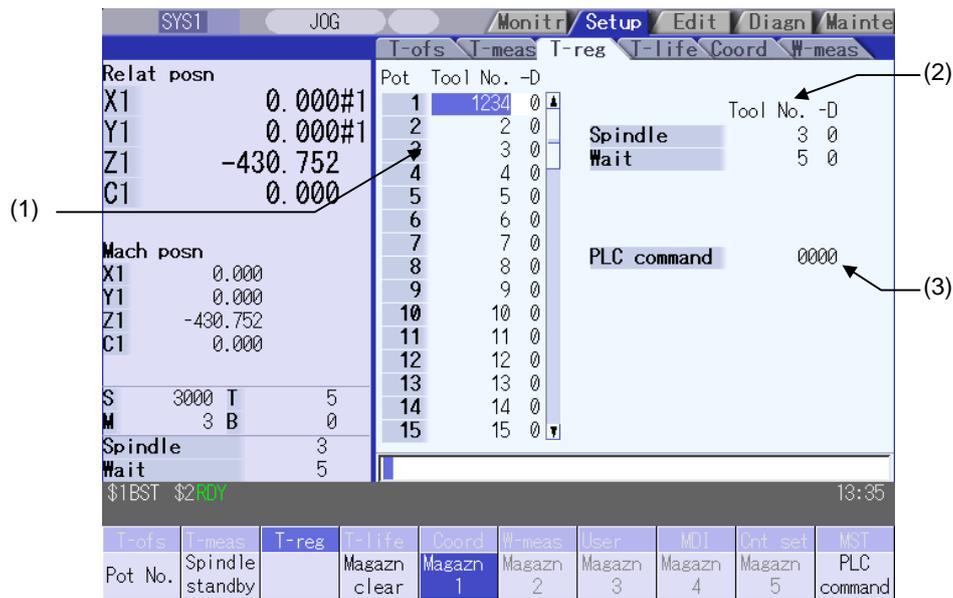


A tool No. is assigned to each tool to make the tools installed on the machine recognizable to the NC. The tool No. is registered corresponding to the magazine pot and spindle where that tool is installed, and the standby location.

When the tool No. and magazine pot are changed by a tool selection command or tool replacement command, the new tool No. is displayed

When not making an arbitrary setting for the number of registered tools, there are maximum 3 magazines, and up to 120 tools can be registered for each magazines. When making an arbitrary setting, there are a maximum of 5 magazines, and up to 360 tools can be registered for all magazines.

This function differs depending on the specifications of the machine tool builder. Refer to the instruction manual issued by the machine tool builder for details.



Inputting an address key, such as MST, displays manual numerical value command window where manual numerical value command is executed.

Display items

Display item	Details
(1) Magazine	This displays the currently selected magazine No.
(2) Tool No.-D	A maximum of 8 digits of data can be input for a tool No. (Refer to the instruction manual issued by the machine tool builder for D functions and purposes.) When the "Tool No." column is blank, confirm the tool first, since the tool could not be mounted, or the tool could not match the spindle. Setting "0" erases the registered tool.
(3) Spindle/Wait (used by PLC program)	This displays the magazine 1 spindle or standby tool No. (The name of this area differs depending on the output from the PLC program.) (Note) Displayed contents depend on the ATC parameter setting.
(4) PLC command	This command is used to input data and perform sequence processing using the PLC program.

Menus

Menu	Details	Type	Reference
Pot No.	Set the magazine pot number and press the INPUT key to position that pot number at the top and display the tool number. The cursor moves to the tool number for that pot number.	A	3.4.1 Registering a Tool in the Magazine Pot
Spindle standby	This sets or erases the tool No. of spindle and standby tools. The cursor will move to the top tool No. of the spindle and standby tools and the setting mode will be entered.	A	3.4.3 Setting/Erasing the Tool No. of Spindle/Standby Tools
Magazn clear	This erases all the currently selected magazine tool data (tool number and D). (Note 1) Unselected magazine tool data is not erased. (Note 2) The spindle and standby data cannot be erased.	A	3.4.3 Setting/Erasing the Tool No. of Spindle/Standby Tools
Magazn 1 : : Magazn 5	This designates the magazine No. to be displayed on the screen. The displayed menu differs according to the machine specifications. [Ex.] When there are two magazines, only the menu Magazine 1 and Magazine 2 keys display.	B	3.4.1 Registering a Tool in the Magazine Pot
PLC command	The cursor moves to the PLC command setting area and the mode changes to the setting mode.	B	3.4.2 Setting the PLC Command

Tool registration duplication check

When the tool No. is set to the magazine pot and the spindle/standby, tool registration duplication can be checked. Either of "Executing the duplication check for all valid magazines", "The duplication check invalid" or "Executing the duplication check only in the selected magazine" can be selected by the parameter "#8922 T-reg-no dup check".

<Priority of duplication check>

Operation \ Parameter	Setting of the operation parameter "T-reg-no dup check"		
	0:Executing the duplication check for all valid magazines	1:The duplication check invalid	2:Executing the duplication check only in the selected magazine
Register tool to magazine pot (Refer to 3.4.1 Registering a Tool in the Magazine Pot)	1. Magazine n (n =1,2...) 2. Spindle/standby	The duplication check is not executed.	1. Selected magazine 2. Spindle/standby
Set tool No. to spindle/standby (Refer to 3.4.3 Setting/Erasing the Tool No. of Spindle/Standby Tools)	1. Spindle/standby 2. Magazine n (n =1,2...)	The duplication check is not executed.	1. Spindle/standby 2. Selected magazine

3.4.1 Registering a Tool in the Magazine Pot

Operation method (Selecting a magazine No.)

- (1) Press the menu .



The tool data of the set magazine No. appears.

Pot	Tool No.	-D
1	12345678	32
2	2	0
3	3	0

(Note) The No. of magazines differs according to the machine specification.

Operation method (Registering the tool in the magazine pot)

- (1) Designate the pot No.



The tool data appears with the designated pot No. at the head.

The cursor moves to the tool No. setting column.

Pot	Tool No.	-D
21	5678	2
22	2	0
23	3	0

- (2) Input the tool number.



The input tool number displays and the cursor moves to the next tool number.

Pot	Tool No.	-D
21	50	2
22	2	0
23	3	0

When the key is pressed without setting a value, the tool number does not change and the cursor moves to the "tool number" for the next pot number.

When the duplication check is valid, the operation message "Exists in magazine 1. Set? (Y/N)" or "Exists in spindle/standby. Set? (Y/N)", etc. appears if the input tool No. already exists in valid magazine, spindle/standby.

(Note 1) To set "D", use the key to move the cursor.

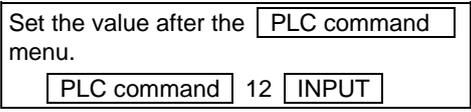
Refer to the manual issued by the machine tool builder for details on the "D" functions and purposes.

(Note 2) The duplication check is executed for only the tool No. The check does not depend on the setting of "D".

(Note 3) If parameter "#8922 T-reg-no dup check" is set to "1", the operation message will not appear even when the tool No. duplicates.

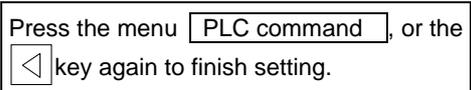
3.4.2 Setting the PLC Command

Operation method (Setting the PLC command)

- (1) Set the value after the **PLC command** menu.  → The set value displays in the PLC command setting field, and the PLC command setting mode is enabled. The function of the command depends on the machine tool builder's specifications.

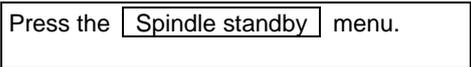
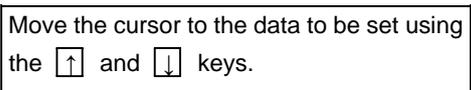
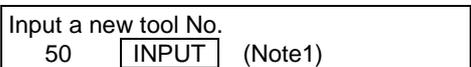
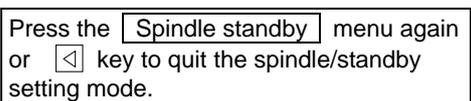
PLC command 0012

(Note) Select the **PLC Command** menu again before pressing the **INPUT** key, then the menu highlight returns to normal, and the PLC command setting mode becomes disable.

- (2) Press the menu **PLC command**, or the  key again to finish setting.  → The PLC command setting mode is released, and the menu highlight returns to normal.

3.4.3 Setting/Erasing the Tool No. of Spindle/Standby Tools

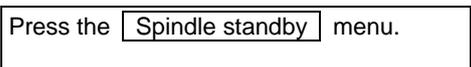
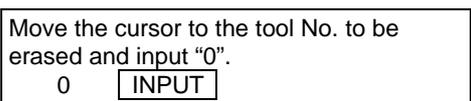
Operation method (Setting the tool No. of spindle/standby tools)

- (1) Press the **Spindle standby** menu.  → The cursor appears at the top tool No. of spindle/standby. In addition, the first "D" column is highlighted.
- (2) Move the cursor to the data to be set using the  and  keys. 
- (3) Input a new tool No. 50 **INPUT** (Note1)  → The tool No. changes. The cursor moves to the next tool No.
- (4) Press the **Spindle standby** menu again or  key to quit the spindle/standby setting mode.  → The spindle/standby setting mode is cancelled.

(Note 1) When the **Spindle standby** menu again or  key is pressed before pressing the **INPUT** key, the spindle/standby setting mode will be cancelled and the set data will be ignored.

(Note 2) Move the cursor with  key to set the "D" data.

Operation method (Erasing the tool No. of spindle/standby tools)

- (1) Press the **Spindle standby** menu.  → The cursor appears at the top tool No. of spindle/standby. In addition, the first "D" column is highlighted.
- (2) Move the cursor to the tool No. to be erased and input "0".  → The tool No. of spindle/standby changes to "0".

(Note 1) Move the cursor with  key and set "0" to erase the "D" data as the same manner as the tool No.

3.4.4 Erasing the Tool Registration Data

Operation method (Selecting a magazine number and erasing the tool registration data.)

- (1) Press the menu .  The tool registration data of the selected magazine No. appears.
- (2) Press the menu .  A message displays confirming whether it is okay to erase.
- (3) Press the or key.  All selected magazine tool data (tool number and D) is erased.
Spindle/standby tool data is not erased.



3.5 Tool Life Management

The life management data of the tool usage conditions, etc., is set and displayed.

The tool life management specifications differ for the M and L system.

Inputting an address key, such as MST, displays manual numerical value command window where manual numerical value command is executed.

<M system>

■ Tool life management I

The usage time or No. of uses of the tool commanded in the program is incremented, and the usage state of that tool is monitored.

■ Tool life management II

This method is the same as tool life management I, but with a spare tool selection function added.

A spare tool is selected from the group of tool commands commanded in the program. Tool compensation (tool length compensation and tool radius compensation) is carried out for the selected tool.

The tool life management related parameters (basic common parameters) include the following.

#	Item	Setting	Details
1103	T_Life T-life manage valid	0	This ignores the tool life management data.
		1	This controls the tool life management.
1104	T_Com2 Tool com 2 (When tool life management is valid.)	0	This treats the program tool command as a group No. Search for the group No. that matches with the tool No. in the tool registration data, and select the spare tool from there.
		1	This treats the program tool command as a tool No.
1105	T_sel2 Tool com 2 (When tool life management is valid.)	0	This selects the tool from the tools in use in the same group, following the registration No. order. If there are no "Tools in use", the tools are selected in order of "Tools not in use", "Normal life tools" and "Abnormal tools", following the registration No. order.
		1	This selects the tool with the maximum remaining life from the tools in use and not in use in the same group. When several tools have the same remaining life, the tools are selected in order of registration No. If there are no "Tools in use" or "Tools not in use", the tools are selected in order of "Tools not in use", "Normal life tools" and "Abnormal tools", following the registration No. order.

<L system>

■ Tool life management I Tool life data display

The tool usage time and usage count indicated in the program is accumulated, and the usage status for that tool is monitored. Tool life management can be performed for a maximum of 80 tools (tool Nos. 1 to 80).

(a) Time based control

The cutting time (G01, G02, G33 etc.) after performing the tool selection command (T) is calculated in the tool usage time corresponding to the designated tool.

A warning is issued if the usage time at the time the tool selection command is performed reaches the life time.

(b) Usage count based control

The tool usage count corresponding to the designated tool No. increases each time the tool selection command (T) is performed.

A warning is issued if the usage count at the time the tool selection command is performed exceeds the life time.

■ Tool life management II Tool life data display / registration group list display

The life (usage time, usage count) of each tool is controlled, and if the life is reached, the same spare tools are selected and used one by one from the group to which that tool belongs.

- Group count : Multi-part system spec. : max. 40 groups for each part system
Single part system spec.: max. 80 groups

- No. of tools in group : Max. 16

The tool life management related parameters (basic shared parameters) are shown below.

#	Name	Setting	Details
1096	T_Ltyp (For L system only) Tool life management type	1	Tool life management I
		2	Tool life management II
1103	T_Life Enable tool life management	0	The tool life management is not performed.
		1	The tool life management is performed.
1107	TlIfsc (For L system only) Tool life management Display screen division	Set the no. of groups displayed at the tool life management II (L series) screen.	
		0	Displayed no. of groups: 1, Max. No. of registered tools: 16
		1	Displayed no. of groups: 2, Max. No. of registered tools: 8
		2	Displayed no. of groups: 4, Max. No. of registered tools: 4

The tool life management screen is made up of the group list display screen and the life management data screen.

(There is no group list display screen in L system tool life management 1. Only the life management data screen exists.)

Refer to the following sections for details of the display contents, menus, and an explanation of the operations for each screen.

"3.5.1 Displaying the Group List"

"3.5.2 Displaying the Life Management Data in Group Units (M system)"

"3.5.3 Displaying the Life Management Data (L system: Tool life management I)"

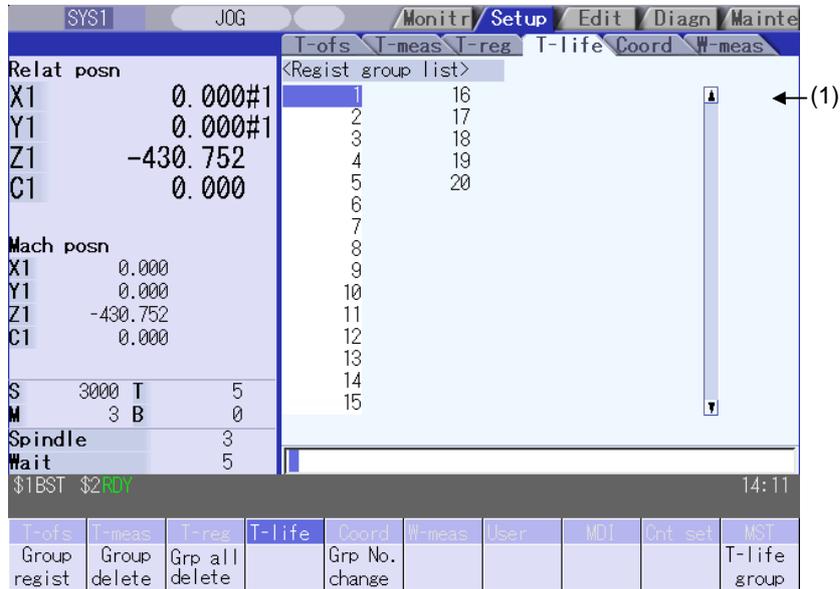
"3.5.4 Displaying the Tool Life Management Data in Group Units (L system: Tool life management II)"

3.5.1 Displaying the Group List

Tool life management data groups can be registered and erased.

<M system>

When changing from a group unit display to the group list display, the cursor displays at the group No. at the group unit display.



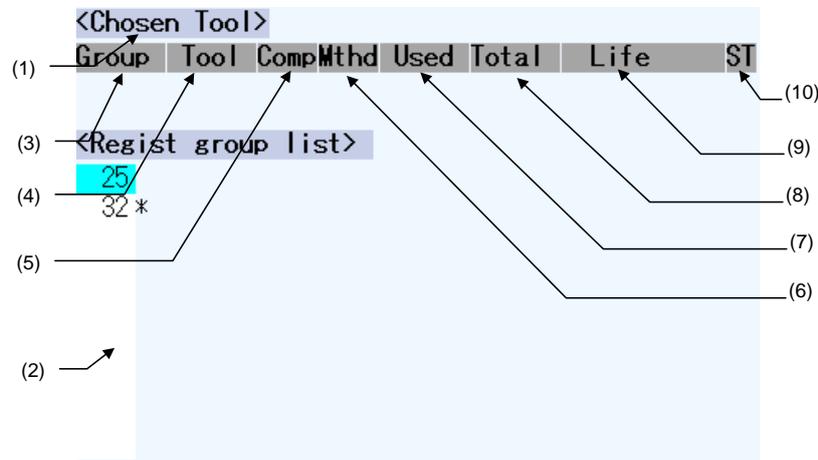
Display items

(M system)

Display item	Details
(1) Regist group list	<p>The group Nos. registered as the tool life management data are listed in ascending order.</p> <p>Group Nos. can be set from 1 to 99999999, and maximum 1000 groups ^(Note) can be set.</p> <p>Maximum 60 groups can be displayed at a time. Use the  /  keys to change the display if the No. of registered groups exceeds 60.</p> <p>An asterisk will be added to the No. of the group in which all tools have reached their lives.</p> <p>(Note) The maximum number of the registered groups depends on the option.</p>

<L system>

This screen is enabled only for tool life management type II (parameter "#1096 T_Ltyp" = 2).
The tool life management data currently being used and the group list display.



Display items

(L system)

Display item	Details
(1) Chosen Tool	The life management information of the tool currently being used appears here.
(2) Regist group list	The group Nos. registered as the tool life management data are displayed in a list. Group Nos. can be set from 1 to 9999, and maximum 80 groups can be set. An asterisk will be added to the No. of the group in which all tools have reached their lives. (Note) The maximum number of the registered groups depends on the option.

[CHOSEN TOOL]

Display item	Details	Display range
(3) Group	Displays the life management group No. currently being used.	1 to 9999
(4) Tool No.	Displays the tool No. currently being used.	1 to 999999
(5) Comp No.	Displays the compensation No. currently being used.	1 to 80
(6) Mthd	Displays whether the group currently being used is controlled by the usage time or usage count.	0: Time 1: Count
(7) Used	Displays the usage data for the tool currently being used.	Usage time: 0 to 99999999 (s) Usage count: 0 to 999999 (set)
(8) Total	Displays the total usage data for the tool currently being used. In the case of tools using multiple compensation Nos., the total usage data for all compensation Nos. displays. In the case of only one compensation No., the same value as "Used" displays.	Usage time: 0 to 99999999 (s) Usage count: 0 to 999999 (set)
(9) Life	Displays the life value for the tool currently being used.	Usage time: 0 to 99999999 (s) Usage count: 0 to 999999 (set)
(10) ST	Displays the status of the tool currently being used.	0: Unused tool 1: Used tool 2: Normal life tool 3: Skipped tool

Menus (M system/L system)

Menu	Details	Type	Reference
Group regist	Creates a new group and adds the group No. to the list.	A	"Registering a group"
Group delete	This erases all of the tool life management data included in the designated group No. If the group No. is not designated, the tool life management data for the group No. indicated by the cursor will be erased.	A	"Erasing a group"
Grp all delete	This erases all groups and their tool life management data registered in the part system which is currently selected.	A	"Erasing all groups"
Group change	Changes the group No.. Specify a new group No. and press the <input type="text" value="INPUT"/> key to change the group No. and then change the contents of the list. An error occurs if an existing group No. is specified.	A	"Changes the group No."
T-life group	The tool life data for the group No. indicated by the cursor will be displayed. This changes to the mode enabling tool life management data to be set and displayed with group unit Nos.	C	"Changing to the group unit display"

Operation method (Registering a group) (M system/L system)

- (1) Press the menu .
- (2) Designate the group No. to be registered.  The life management data for the selected group No. displays empty.
 The group No., method, and life value is set at the L system.
 (Example) 25/1/3000

(Note 1) An error occurs if the group No. duplicates a pre-existing group No.

(Note 2) The group is not registered if the following operations are performed after newly registering a group No..

- The application is exited without registering even 1 item of tool life data.
- The screen is changed to that of other than the group unit display.
- The part system is changed.

Operation method (Erasing a group) (M system/L system)

- (1) Press the menu .
- (2) Designate the group to be erased.  The selected group No. is erased.

If nothing is input in the input area, the group at the cursor location is erased.

Operation method (Erasing all groups)

(M system/L system)

- (1) Press the menu . → The confirmation message appears.
- (2) Designate the group to be erased.
72 → All the registered groups and their tool life management data is erased.

(Note 1) If no group has been registered, the operation message "Can't delete all groups" will appears.

Operation method (Changing the group No.)

(M system/L system)

(Ex.) Changing the group No. from 5 to 20

- (1) Use the , , and keys to move the cursor to group No. 5.
- (2) Press the menu key. → The menu is highlighted.
- (3) Set the new group No..
20 → The group No. changes to "20".

Operation method (Changing to the group unit display)

(M system/L system)

(Ex.) Displaying the group No. 5 tool life data.

- (1) Use the , , and keys to move the cursor to group No. 5.
- (2) Press the menu or keys. → The screen changes to the group unit display and the group No. "5" tool life management data displays.

<M system>

Group No.	5					
#	Tool No.	ST	Mthd	L compen	R compen	
1	301	00	000	0	0	
2	302	00	000	0	0	
3	303	00	000	0	0	
4	304	00	000	0	0	
5	305	00	000	0	0	

<L system>

Group:	5 Form: 1 Life: 3000 (set)				
#	Tool	Comp	Used (set)	ST	
1	1	2	0	0	
2	2	2	2	1	
3	5	5	5	1	
4	6	6	6	1	
5	7	7	7	1	
6	8	8	8	1	
7	9	9	9	1	
8	10	10	10	1	

3.5.2 Displaying the Life Management Data in Group Units (M system)

The tool life management data of an arbitrary group is set and displayed. If the registered tools cannot be displayed on one screen, scroll the screen using the  or  key.

Group No.	Tool No.	ST Mthd	L compen	R compen
1	101	01 220	80.000	8.000
2	102	00 220	82.000	8.000
3	103	00 220	88.000	8.000
4	104	00 220	90.000	8.000
5	105	00 220	100.000	8.000

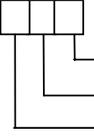
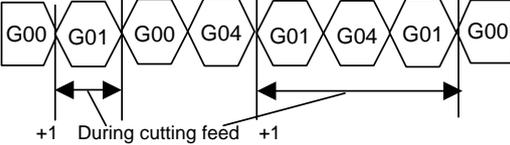
Group No.	Tool No.	Auxil	Life Used
1	101	1	120 30 min
2	102	0	120 0 min
3	103	0	120 0 min
4	104	0	80 0 min
5	105	0	100 0 min

Display items

(M system)

Display item	Details	Display range
(1) Group No.	This is the group No. of the tool for which tool life management is performed. Tools with the same group No. are regarded as spare tools.	1 to 99999999
(2) #	This is the data setting No.. This is not the magazine pot No.	
(3) Tool No.	This is the No. corresponding to the each individual tool. Maximum 1000 tools can be registered. This is a fixed tool No. actually output for the tool command and so on.	1 to 99999999
(4) ST	 <p>Tool status Machine manufacturer release</p> <p>Tool status 0 : Unused tool Normally set to "0" when the tool is replaced with a new tool. 1 : Used tool This becomes "1" when the cutting actually starts. 2 : Normal life tool This becomes "2" when the usage data (usage time and No. of uses) exceeds the life data. 3 : Tool abnormality 1 tool 4 : Tool abnormality 2 tool</p> <p>(Note) Item 3 and 4 differ depending on the machine tool builder specifications.</p>	

(M system)

Display item	Details	Display range
(5) Mthd	 <p>(a) Tool life management method (b) Tool radius compensation data format (c) Tool length compensation data format</p> <p>(a) Tool life management method 0 : Usage time This manages by the cutting execution time. 1 : No. of mounting times This manages by the No. of times the tool became the spindle tool in tool replacement, etc. However, in the case where the cutting feed command (G01, G02, G03 etc.) is not executed even once after becoming the spindle tool, the No. of times is not counted. 2 : Usage count This is counted when the rapid traverse command (G00 etc.) is changed to the cutting feed command (G01, G02, G03 etc.). However, this is invalid in the case of a rapid traverse or cutting feed command with no movement.</p>  <p>(b) Tool radius compensation data format (c) Tool length compensation data format 0 : Compensation No. The tool life management data compensation data is handled as a compensation No., this No. replaces the compensation No. designated in the process program, and compensation is performed. 1 : Added compensation amount The tool life management data compensation data is handled as the added compensation amount, an addition is made to the compensation amount shown with the compensation No. designated in the process program, and compensation is performed. 2 : Direct compensation amount The tool life management data compensation data is handled as the direct compensation amount, this No. replaces the compensation amount shown for the compensation No. designated in the process program, and compensation is performed.</p>	
(6) L compen	This depends on the data format designated in "Mthd".	Compensation No. : 1 to 400 Added compensation amount Direct Compensation amount (Note 1)
(7) R compen		
(8) Auxil	This depends on the machine tool builder specifications.	0 to 65535
(9) Life	Set the usage time (minutes), attachment count (No. of times attached to the spindle), or usage count (No. of holes drilled) for the life for each tool based on the data format set for "Method". The life is infinite when "0" is set.	Usage time: 0 to 4000 (min) No. of mounting times: 0 to 65000 (set) No. of uses: 0 to 65000 (set)

(M system)		
Display item	Details	Display range
(10) Used	This displays the usage data for individual tools based on the method specified for the tool life management method. (Note) This data is not counted during machine lock, auxiliary function lock, dry run or for a single block.	Usage time: 0 to 4000 (min) No. of mounting times: 0 to 65000 (set) No. of uses: 0 to 65000 (set)

(Note 1) The "Length compensation" and "Radius compensation" calculation/direct compensation amount setting/display range are as follows based on a combination of settings of "#1003 iunit (input unit)" and "#1041 I_inch (initial inch)".

#1003 iunit	#1041 I_inch	Setting/display range
B	0	-999.999 to 999.999
	1	-99.9999 to 99.9999
C	0	-999.9999 to 999.9999
	1	-99.99999 to 99.99999
D	0	-999.99999 to 999.99999
	1	-99.999999 to 99.999999
E	0	-999.999999 to 999.999999
	1	-99.9999999 to 99.9999999

When data without the decimal point is input, a set unit can be specified by the parameter "#8119 Comp. unit switch".

(Note 2) When the part system is changed while displayed on the group unit display, the registered data will not be displayed because the screen enters the "data unregistered" state.
To display the group unit of the changed part system, return to the group list display screen and execute the group unit display again.

Menus

(M system)

Menu	Details	Type	Reference
Group No.	This displays the data of a group when that group No. of tool life management data is set and the <input type="text" value="INPUT"/> key is pressed. The group No. can be referred to in the group list.	A	"Designating a group No."
Line copy	Copies one line of tool life management data where the cursor is located.	C	"Copying/pasting the tool life data"
Line paste	The copied tool life management data is written to the data in the line where the cursor is located. If the copied data is changed after the menu <input type="text" value="Line copy"/> is pressed, the data before changing is written when pasted. The data in the copied line can be pasted as many times as is required until new data is copied.	C	
Line clear	This erases a designated line (multiple lines possible) of tool life management data. The first and the last line No. of the data to be erased is specified with a "/" separating the Nos. If the <input type="text" value="INPUT"/> key is pressed without specifying a line, the data in the line where the cursor is located is erased.	A	"Erasing one line of tool life management data" "Designating and erasing a line"
Group change	This changes a group No. If a new group No. is set and the <input type="text" value="INPUT"/> key pressed, the group No. changes. An error occurs if a pre-existing group No. is set.	A	"Changing a group No."
Horiz scroll	This scrolls the display of the tool life management data to the left and right. The items below display alternately each time the menu is pressed. 1. Tool No., ST, Mthd, L compen, R compen 2. Tool No., Auxil, Life, Used	C	
Prev group	Displays the previous group No. data.	C	
Next group	Displays the next group No. data.	C	
Group list	This displays the list of tool life data groups.	C	
Group regist	This creates a new group.	A	"Registering a group"
Group delete	This erases all tool life management data contained in the currently displayed group No.	A	"Erasing a group"

Operation method (Designating a group No.)**(M system)**

- (1) Press the menu. → The menu is highlighted.
- (2) Designate the group No. to be displayed.
12 → The tool life data of the designated group No. appears, and the cursor moves to the head of the data.

(Note) If a group No. that does not exist is designated, a message appears to confirm the creation of a new No. A new group is created if or is pressed.

Operation method (Changing a group No.)**(M system)**

- (1) Press the menu. → The menu is highlighted.
- (2) Designate the group No. after changing.
112 → The group No. changes to "112".

(Note) An error occurs if the group No. duplicates a pre-existing group No.

Operation method (Setting the tool life data)**(M system)**

- (1) Move the cursor to the data to be set using the , , , , , and keys. → The menu is highlighted.
It is possible to move to the line after the registered line.
Set a tool No. for this line to newly register tool life management data.
- (2) Set a value.
20 → The set value displays.
The cursor moves one position to the right.

(Note 1) The other settings will be invalid when the tool ST (status) setting data are not set.

(Note 2) The same tool cannot be registered in more than one group.

(Note 3) If the tool No. is not set, the other data cannot be set.

(Note 4) Change the "Method" to initialize the related items.

Change the tool life management method (the 1st digit) to clear the "Life" and "Usage", and also change the units.

Change the tool radius compensation data (the 2nd digit) to clear the "Radius Compensation".

Change the tool length compensation data (the 3rd digit) to clear the "Length Compensation".

Figures after the decimal point are also changed in line with the specifications.

Operation method (Copying/pasting the tool life data)

(M system)

(1) Move the cursor to the line to be copied.

(2) Press the menu .



The menu is highlighted.
The background color of the copied data changes to light blue.
(The color changes even when scrolling horizontally.)

(3) Move the cursor to the line where the data is to be pasted.



Group No.		5				
#	Tool No.	ST	Mthd	L compen	R compen	
1	87654321	0	110	-123456.789	-123456.789	
2	101	1	101	123.456	123	
3	102	2	012	100	123.456	
4	103	3	020	1	-123.456	
5						

(4) Press the menu.



The copied tool life management data is written to the data (excluding the tool No.) in the line where the cursor is located.
The background color returns to normal.
The cursor moves to "ST".

(Note 1) Once copied, the data is held until a new data item is copied.

(Note 2) It is not possible to paste into a line for which a tool No. has not been set.

Operation method (Erasing one line of tool life management data)

(M system)

(1) Move the cursor to the line to be erased.

(2) Press the menu and keys.



The menu is highlighted.
A message displays confirming whether it is okay to erase the data.
The background color of the area to be erased changes to light blue.

(3) Press the or key.



Press a key other than or in order not to erase the data.

One line of the selected life management data is erased and the menu highlight returns to normal.
The next line moves up after erasing.

Operation method (Designating and erasing the multiple lines)

(M system)

- (1) Press the menu . → The menu is highlighted.
- (2) Set the range to be erased by marking the first and last # Nos. with a "/", and press key.
122/125 → A message appears to confirm the erasing. The background color of the area to be erased changes to light blue.
- (3) Press the or key. → The tool life data corresponding to the designated # No. is erased, and the menu highlight returns to normal. The next line after the erased range moves up.
Press a key other than or in order not to erase the data.

Operation method (Registering a group)

(M system)

- (1) Press the menu.
- (2) Set the No. of the group to be registered. → A new group is created and the life management data displays containing no data.
25

(Note 1) An error occurs if the same group No. already exists.

(Note 2) The group is not registered if one of the operations below is performed after registering a new group No..

- If the application is exited without registering even 1 item of tool life data.
- If the screen is changed to that of other than the group unit display.
- The part system is changed.

Operation method (Erasing a group)

(M system)

- (1) Move the cursor to the group No. to be erased.
- (2) Press the menu. → A message appears to confirm the erasing.
- (3) Press the or key. → All the tool life management data for the displayed group No. is erased and the previous group No. data displays. Returns to the group list displays if all groups are erased.

3.5.3 Displaying the Life Management Data (L system: Tool life management I)

The tool life management data can be set and displayed. Use the  /  keys to scroll between screens if all of the registered tools cannot be displayed on one screen.

#	[Time]		[Count]		[Status]	
	Used	Life	Used	Life	A	B
1	46:36:12	0: 0	3000	0	0	0
2	46:47: 7	0: 0	3000	0	2	0
3	46:47: 7	0: 0	3000	0	5	0
4	46:47: 7	0: 0	3000	0	6	0
5	46:47: 7	0: 0	3000	0	7	0
6	46:47: 7	0: 0	3000	0	8	0
7	46:47: 7	0: 0	3000	0	9	0
8	46:47: 7	0: 0	3000	0	10	0
9	0:21:51	0: 0	3232	0	-95	12
10	0: 0: 0	0: 0	0	0	0	0
11	0: 0: 0	0: 0	0	0	0	0
12	0: 0: 0	0: 0	0	0	0	0
13	0: 0: 0	0: 0	0	0	0	0

Display items

(L system)

Display item		Details	
(1) #		Tool No. The # No. is highlighted if the usage time reaches the life time or if the usage count exceeds the life count.	
(2) Time	Used	The integrated time the tool is used. This timer value is incremented during cutting.	0:0:0 to 99:59:59 (h:min:s)
	Life	Tool life time Set the service lifetime.	0:0 to 99:59(h:min) (0:0 = no warning given)
(3) Count	Used	The integrated count the tool is used. The count goes up when a tool is selected.	0 to 9999 (times)
	Life	Tool life count Set the service life count.	0 to 9999 (times) (0:0 = no warning given)
(4) Status	A (left side)	The tool life management status is indicated. 0: Not used 1: Current tool (tool being used) 2: Service lifetime (service life count) is exceeded.	0 to 2
	B (right side)	(Used by machine tool builder)	0 to 99

Menus (L system)

Menu	Details	Type	Reference
Line copy	Copies one line of tool life management data where the cursor is located.	C	"Copying/pasting the tool life data"
Line paste	The copied tool life management data is written to the data in the line where the cursor is located. If the copied data is changed after the menu key <code>Line copy</code> is pressed, the data before changing is written when pasted. The copied data can be pasted as many times as is required until new data is copied.	C	
Line clear	This erases a designated line (multiple lines possible) of tool life management data. The first and the last line No. of the data to be erased is specified with a "/" separating the Nos.. If the <code>INPUT</code> key is pressed without specifying a line, the data in the line where the cursor is located is erased.	A	"Erasing one line of tool life management data" "Designating and erasing a line"

Operation method (Setting the tool life data) (L system)

(1) Move the cursor to the data to be set using the `↑`, `↓`, `↕`, `↔`, `←` and `→` keys.

It is possible to move to the line after the registered line.

(2) Set a value.
20 `INPUT`

➡ The set value displays.
The cursor moves one position to the right.

Operation method (Copying/pasting the tool life data) (L system)

(1) Move the cursor to the line to be copied.

(2) Press the menu `Line copy`.

➡ The background color of the copied data changes to light blue.

(3) Move the cursor to the line where the data is to be pasted.

(4) Press the `Line paste` menu.

➡ The copied tool life management data is written to the data in the line where the cursor is located.
The background color returns to normal.

(Note 1) Once copied, the data is held until a new data item is copied.

Operation method (Erasing one line of tool life management data)

(L system)

- (1) Move the cursor to the line to be erased.
- (2) Press the menu and keys. → The menu is highlighted. A message displays confirming whether it is okay to erase the data. The background color of the area to be erased changes to light blue.
- (3) Press the or key. → One line of the selected life management data is erased and the menu highlight returns to normal. The next line moves up after erasing.
Press a key other than or in order not to erase the data.

Operation method (Designating and erasing the multiple lines)

(L system)

- (1) Press the menu . → The menu is highlighted.
- (2) Set the erasing range by marking the first and last # Nos. with a "/", and press key. → A message displays confirming whether it is okay to erase the data. The background color of the area where the data is erased changes to light blue.
122/125
- (3) Press the or key. → The tool life data corresponding to the designated # No. is erased, and the menu highlight returns to normal. The next line after the erased range moves up.
Press a key other than or in order not to erase the data.

(Note) If the key is pressed without setting a # No., the line at the cursor is erased.

3.5.4 Displaying the Tool Life Management Data in Group Units (L system: Tool life Management II)

The tool life management data of an arbitrary group is set and displayed. If the registered tools cannot be displayed on one screen, scroll the screen using the  or  key.

This screen is enabled only for tool life management II ("#1096 T_Ltyp" = 2).

Group: 1 Form: 1 Life: 300 (set)				
#	Tool	Comp	Used (set)	ST
1	101	11	28	1
2	102	12	0	0
3	103	13	0	0
4	104	14	0	2
5	105	15	0	0
6				
7				
8				

By setting parameter "#1107 Tllfsc", it is possible to select the life management data display mode for multiple groups.

"#1107 Tllfsc" setting value	0	1	2
No. of display groups	1	2	4
Max. No. of registered tools	16	8	4

<Group 1 display mode> (#1107 Tllfsc = 0)

Group: 1 Form: 1 Life: 300 (set)					Group: 9 Form: 1 Life: 300 (set)				
#	Tool	Comp	Used (set)	ST	#	Tool	Comp	Used (set)	ST
1	101	11	28	1	9				
2	102	12	0	0	10				
3	103	13	0	0	11				
4	104	14	0	2	12				
5	105	15	0	0	13				
6					14				
7					15				
8					16				

<Group 2 display mode> (#1107 Tllfsc = 1)

Group: 1 Form: 1 Life: 300 (set)					Group: 2 Form: 1 Life: 200 (set)				
#	Tool	Comp	Used (set)	ST	#	Tool	Comp	Used (set)	ST
1	101	11	28	1	1	201	21	11	1
2	102	12	0	0	2	202	22	0	0
3	103	13	0	0	3	203	23	0	0
4	104	14	0	2	4				
5	105	15	0	0	5				
6					6				
7					7				
8					8				

<Group 4 display mode> (#1107 Tllfsc = 2)

Group: 1 Form:1Life: 300 (set)					Group: 2 Form:1Life: 200 (set)				
#	Tool	Comp	Used (set)	ST	#	Tool	Comp	Used (set)	ST
1	101	11	28	1	1	201	21	11	1
2	102	12	0	0	2	202	22	0	0
3	103	13	0	0	3	203	23	0	0
4	104	14	0	2	4				
Group: 3 Form:0Life: 30 s					Group: 4 Form:0Life: 50 s				
#	Tool	Comp	Used (s)	ST	#	Tool	Comp	Used (s)	ST
1	301	31	5	1	1	401	41	5	1
2	302	32	0	0	2	402	42	0	0
3	303	33	0	0	3	403	43	0	0
4					4				

Display items

(L system)

<Group information>

Display item	Details	Setting range
(1) Group	Life management group No.	1 to 9999
(2) Form	This displays whether to control the currently displayed group in usage hours or usage count.	0 : Hours 1 : Times
(3) Life	This sets the tool life data for the currently displayed group. The following tool life data units display based on the control method. Usage time : (s) Usage count : (set)	Usage time: 0 to 99999999 (s) Usage count: 0 to 999999 (set)

<Life management data within group>

Display item	Details	Setting range
(4) #	This displays whether the tool for which the # No. is highlighted is a life tool (ST=2) or a skip tool (ST=3).	
(5) Tool No.	This sets the tool No.	1 to 999999
(6) Comp No.	This sets the compensation No.	1 to 80
(7) Used	This sets the usage data at such times as when the tool is not an unused tool. This becomes "0" when the usage data is not set. The following usage data units display based on the control method. Usage time : (s) Usage count : (set)	Usage time: 0 to 99999999 (s) Usage count: 0 to 999999 (set) Setting can be omitted.
(8) ST	This sets the tool status. This becomes "0" when the tool status is not set. 0: Unused tool 1: Used tool 2: Normal life tool 3: Skipped tool	0 to 3 Setting can be omitted.

Menus (L system)

Menu	Details	Type	Reference
Group No.	This displays the data of a group when that group No. of tool life management data is set and the <input type="text" value="INPUT"/> key is pressed. The group No. can be referred to in the group list.	A	"Designating a group No."
Line copy	Copies one line of tool life management data where the cursor is located.	C	"Copying/pasting the tool life data"
Line paste	The copied tool life management data is written to the data in the line where the cursor is located. If the copied data is changed after the menu key <input type="text" value="Line copy"/> is pressed, the data before changing is written when pasted. The copied data can be pasted as many times as is required until new data is copied.	B	
Line clear	This erases a designated line (multiple lines possible) of tool life management data. The first line No. and the last line No. of the data to be erased is specified with a "/" separating the Nos. If the <input type="text" value="INPUT"/> key is pressed without specifying a line, the data in the line where the cursor is located is erased.	A	"Erasing one line of tool life management data" "Designating and erasing a line"
Grp No. change	This changes a group No. If a new group No. is specified and the <input type="text" value="INPUT"/> key pressed, the group No. changes. An error occurs if a pre-existing group No. is set.	A	"Changing a group No."
Group change	The group active area is changed if parameter "#1107 TlIfsc" is set to 1 or 2.	C	"Changing the active area for multiple group displays"
Multi group	Changes to the multiple group life management data display mode based on the parameter "#1107 TlIfsc" setting value.	C	
Group regist	This creates a new group and displays the life management data containing no data.	A	"Registering a group"
Group list	This displays the list of tool life data groups.	C	
Close	The multiple group life management data display mode is cancelled.	C	

Operation method (Designating a group No.) (L system)

- (1) menu. The menu is highlighted.
- (2) Designate the group No. to be displayed. The tool life data of the designated group No. appears, and the cursor moves to the head of the data.

(Note) If a group No. that does not exist is set, a message appears to confirm the creation of a new No.. A new group is created if or is pressed.
When creating a new group, set the method and life data at the end of the group No., separating the data with a "/". If the method and life data are omitted, a new group is created with method = 0 (usage time), and life data = 0.

Operation method (Changing a group No.)

(L system)

- (1) Press the menu. → The menu is highlighted.
- (2) Designate the group No. after changing.
112 → The group No. changes to "112".

(Note) An error occurs if the group No. duplicates a pre-existing group No.

Operation method (Setting the tool life data)

(L system)

- (1) Move the cursor to the data to be set using the , , , , and keys.
It is possible to move to the line after the registered line.
Set a tool No. for this line to newly register tool life management data.
- (2) Set a value.
20 → The set value displays.
The cursor moves one position to the right.

(Note 1) It is not possible to set any other data if the tool No. and compensation No. have not been set.

Operation method (Copying/pasting the tool life data)

(L system)

- (1) Move the cursor to the line to be copied.
- (2) Press the menu . → The background color of the copied data changes to light blue.
- (3) Move the cursor to the line where the data is to be pasted.
- (4) Press the menu. → The copied tool life management data is written to the data in the line where the cursor is located.
The background color returns to normal.
The cursor moves to "Tool No.".

(Note 1) Once copied, the data is held until a new data item is copied.

Operation method (Erasing one line of tool life management data)

(L system)

- (1) Move the cursor to the line to be erased.
- (2) Press the menu and keys. → The menu is highlighted. A message displays confirming whether it is okay to erase the data. The background color of the area where the data is erased changes to light blue.
- (3) Press the or key. → One line of the selected life management data is erased and the menu highlight returns to normal. The next line moves up after erasing.
Press a key other than or in order not to erase the data.

Operation method (Designating and erasing the multiple lines)

(L system)

- (1) Press the menu .
- (2) Set the erasing range by marking the first and last # Nos. with a "/", and press key. → A message displays confirming whether it is okay to erase the data. The background color of the area where the data is erased changes to light blue.
- (3) Press the or key. → The tool life data corresponding to the designated # No. is erased, and the menu highlight returns to normal. The next line after the erased range moves up.
Press a key other than or in order not to erase the data.

(Note) If the key is pressed without setting a # No., the line at the cursor is erased.

Operation method (Registering a group)

(L system)

- (1) Press the menu.
- (2) Set the No. of the group to be registered. → A new group is created and the life management data displays containing no data.

(Note 1) If the method and life data are omitted, a new group is created with method = 0 (usage time), and life data = 0.

(Note 2) An error occurs if the same group No. already exists.

(Note 3) The group is not registered if one of the operations below is performed after registering a new group No.

- If the application is exited without registering even 1 item of tool life data.
- If the screen is changed to that of other than the group unit display.
- The part system is changed.

Operation method (Changing the active area for multiple group displays)

(L system)

(1) Press the Group change menu.



The cursor indicating the active status moves to the next group.

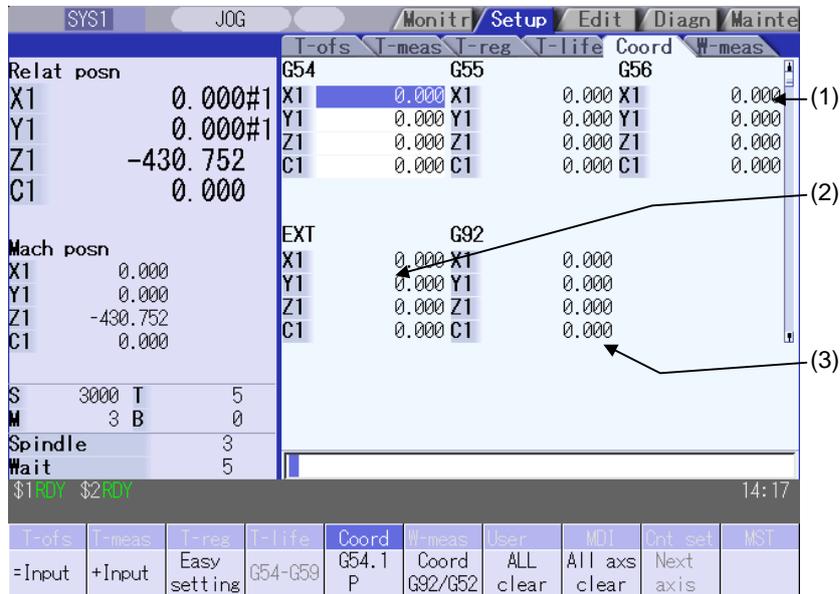
Group: 1 Form: 1 Life: 300 (set)					Group: 2 Form: 1 Life: 200 (set)				
#	Tool	Comp	Used (set)	ST	#	Tool	Comp	Used (set)	ST
1	101	11	28	1	1	201	21	11	1
2	102	12	0	0	2	202	22	0	0
3	103	13	0	0	3	203	23	0	0
4	104	14	0	2	4				
5	105	15	0	0	5				
6					6				
7					7				
8					8				

3.6 Workpiece Coordinate System Offset



This function allows the user to set and display the coordinate system offset controlled by the NC. Using the option, the number of coordinate system offset sets can be increased by 48 or 96 sets.

(Note) The G92/G52 coordinate system offset cannot be set.



Inputting an address key, such as MST, displays manual numerical value command window where manual numerical value command is executed.

Display items

Display item	Details
(1) Coordinate system offset area	<p>This sets and displays the offset amount for the workpiece coordinate system (G54 - G59), or the extension workpiece coordinate system (G54.1Pn). Use the menu operation or page change key to specify which workpiece coordinate system offset to display or set. The workpiece coordinate system offset amount data can be set in absolute or additional mode.</p>
(2) EXT offset area	This displays or sets the offset amount for the external workpiece system.
(3) G92/G52 offset area	This displays the offset amount for G92 or the local coordinate system. The offset amount of the corresponding local coordinate system (G52) is displayed only when the cursor is at G54 to G59.

CAUTION

If the tool compensation amount or workpiece coordinate system offset amount is changed during automatic operation (including during single block stop), the changes will be valid from the command in next block or after several subsequent blocks.

Menus

Menu	Details	Type	Reference
= Input	This inputs the offset amount with the absolute mode.	C	3.6.1 Setting the Coordinate System Offset
+ Input	This inputs the offset amount with the additional mode.	C	
Easy setting	This sets the workpiece coordinate system offset so that the current machine coordinate becomes the workpiece coordinate zero point. Only the axis at the cursor position is set. Easy setting cannot be executed for an axis of auxiliary axis state.	C	3.6.3 Setting the Work Piece Coordinate Origin
G54-G59	This displays the G54 - G59 workpiece coordinate system offset amount. The cursor moves to the G54 offset. This menu can be used when the G54 – G59 offsets are not displayed.	C	3.6.1 Setting the Coordinate System Offset
G54.1 P	Enter the P number displayed at the screen to display the extension workpiece coordinate (G54.1 Pn) offset in the coordinate system offset area. This menu is not displayed if the extension workpiece coordinate system offset option is disabled. If the local coordinate system offset (G52) displays in the G92/G52 area, the display changes to G92.	A	
Coord G92/G52	This displays the G92 or G52 coordinate system offset amount. The G92 offset data is always displayed when the cursor is not at the G54 to G59 offset data.	C	
All clear	All coordinate system offset amounts for all axes are erased with the exception of G92 and EXT.	A	3.6.2 Erasing the Coordinate System Offset Amount
All axis clear	This erases the local offset data for all axes corresponding to the offset data of the coordinate system where the cursor is located.	A	
Next axis	This can be selected when the number of enabled axes is 6 or over. The displayed axes are changed to axes 1 – 5, and axis 6 and over. (Note) This displays when there are 6 or more axes.	C	

3.6.1 Setting the Coordinate System Offset

Operation method (Setting the workpiece coordinate system G54 - G59 offset amount)

- (1) Press the menu.  The G54 - G59 workpiece coordinate system offset displays.
- (2) Use the (,) keys to move the cursor to the workpiece coordinate system offset to be set.
Use the , page keys to change the coordinate system display.
Refer to the "Changing the offset amount using the page change keys (previous page: , next page:)" section in "3.6.4 Changing the coordinate system display" for further details.
- (3) Use the , keys to move the cursor to the axis for which data is to be set.
- (4) Set the offset amount.
122  The set data displays and the cursor moves to the next data.
This setting can be made by pressing the menu key instead of the key.
- (5) Set other data using the same method.

(Note) It is possible to perform additional input by pressing the menu key instead of the or menu keys.

[Ex.] If the original data is 122.000:

$$1.234 [+INPUT] \rightarrow 122.000 + 1.234 = 123.234$$

Setting the external coordinate system offset amount

Set the external coordinate system offset amount to keep the current workpiece coordinate system offset amounts from G54 to G59, and also apply an offset to the entire system.

Furthermore, it is also possible to use the data transfer function to write data directly to the external coordinate system offset (EXT).

(Refer to "6.2 Input/output screen" for details of the data transfer function.)

3.6.2 Erasing the Coordinate System Offset Amount

Operation method (Erasing the coordinate system (excluding G92) offset amount where the cursor is displayed for all axes.)

- (1) Press the menu menu.  The menu is highlighted.
A message displays confirming whether it is okay to erase the data.
- (2) Press the key.  All of the coordinate system indicated by the cursor is cleared to zero.
In addition, all axes data of the local offset (G52) for the coordinate system will be erased.

Operation method (Erasing all axis offset data of all coordinate systems (excluding G92 and EXT))

- (1) Press the menu menu.  The menu is highlighted.
A message displays confirming whether it is okay to erase the data.
- (2) Press the key.  All coordinate system data is erased.
In addition, all axes data of the local offset (G52) for the G54 to G59 coordinate systems will be erased.

(Note) G92 data cannot be erased.

3.6.3 Setting the Workpiece Coordinate Origin

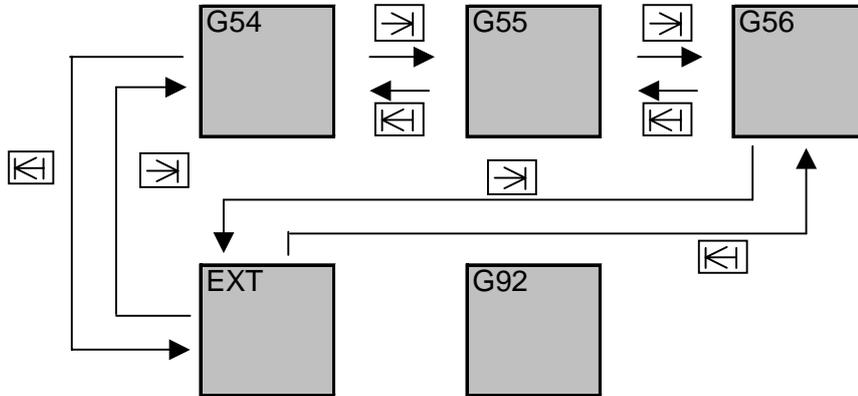
Press the menu key to set the coordinate system offset so that the current machine position (for all axes) becomes the workpiece coordinate origin. It is only possible to set those axes where the cursor is located.

If the menu is pressed when there is the cursor in coordinate system offset (G54 to G59), the local offset (G52) data of the axis will be also erased.

3.6.4 Changing the Coordinate System Display

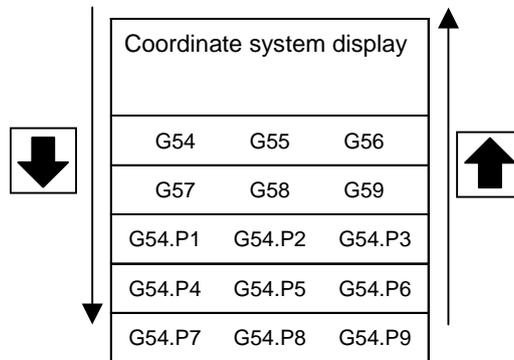
Use the   tab keys to change the coordinate system offset cursor position.

Press the Tab key to change the cursor position.



Changing the coordinate system offset using the page change keys (previous page: , next page: )

Use the page change keys to change the coordinate system offset display as shown below.



An option is required for the G54.P1 - P96 workpiece coordinate system offset display.



3.7 Workpiece Measurement

3.7.1 Workpiece Measurement (M system)

Move the sensor attached to the spindle using manual feed or handle feed to contact the workpiece, measure the coordinate position, and then set that measurement result for the workpiece coordinate system offset amount.

There are three measurement patterns: surface measurement, hole measurement, width measurement, and rotation measurement.

For the rotation measurement, the measurement result is set in the workpiece offset (rotation center) and parameters "#8624 Coord rot centr(V)", "#8626 Coord rot vctr(H)", and "#8627 Coord rot angle".

■ For surface measurement (As for the hole or width measurement, the guide drawing differ.)

(1) Relat posn
X1 1569.122
Y1 -430.368
Z1 -367.299

(2) T-ofs T-meas T-reg T-life Coord W-meas

(3) Face meas On mea
X -1566.122
Y -433.368
Z

(4) Mach posn
X1 -1569.122
Y1 -430.368
Z1 -367.299

(5) S 300 T 0
M 0 B 0

(6) Sensor L 100.000
Sensor D 6.000
G54 Meas posn
X1 0.000 X1 -1566.122
Y1 0.000 Y1 -433.368
Z1 0.000 Z1 -467.299

(7) 16:02

(8) T-ofs T-meas T-reg T-life Coord W-meas User MDI Ont set MST
Coord write Coord 654-659 Coord 654.1P_ Coord EXT SkipPos take in Face meas Hole meas Width meas Rotate measure Next axis

■ For rotation measurement

(1) Relat posn
X1 -1548.974
Y1 -400.738
Z1 -367.299

(2) T-ofs T-meas T-reg T-life Coord W-meas

(3) Rotate meas On mea
A:X1 -1548.974
Y1 -400.738
B:X1
Y1
C:X1
Y1

(4) C shift (s)
X1 0.000
Y1 0.000

(5) G54 Coord center Meas posn
X1 0.000 X1 -1551.974
Y1 0.000 Y1 -397.738
Z1 0.000 Coord angle Z1 -467.299
0.000

(6) Sensor L 100.000
Sensor D 6.000

(7) 16:07

(8) T-ofs T-meas T-reg T-life Coord W-meas User MDI Ont set MST
Coord write Coord 654-659 Coord 654.1P_ Coord EXT SkipPos take in Face meas Hole meas Width meas Rotate measure Next axis

Inputting an address key, such as MST, displays manual numerical value command window where manual numerical value command is executed.

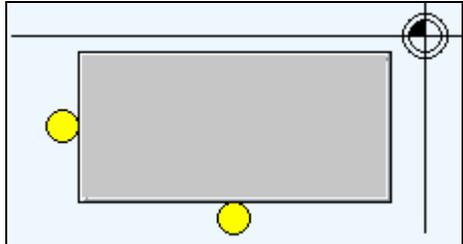
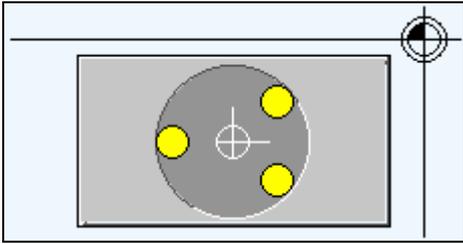
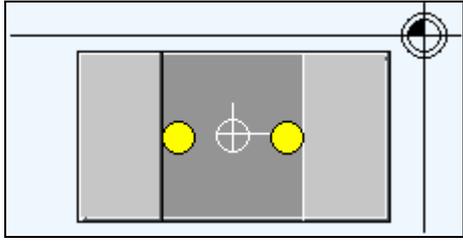
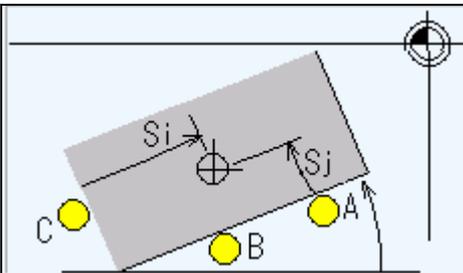
Display items

Display item	Details												
(1) Measurement counter	<p>Displays the measurement position. The measurement axis is compatible with base system parameters "#1026 base_I", "#1027 base_J", and "#1028 base_K". (Hereafter indicated as I=X, J=Y, and K=Z axes in this manual.) The measurement position becomes blank when setting the coordinate system offset amount, resetting the NC, or changing the measurement mode (when face meas, hole meas, width meas, or rotate meas menu is selected). The display contents differ depending on the measurement pattern (surface measurement, hole measurement, width measurement, or rotation measurement).</p>												
Surface measurement counter	Displays the calculated measurement position from the skip position for each axis (X-, Y-, Z-axes).												
Hole measurement counter	Displays 3 (A, B, C) measurement positions (X-, Y-axes).												
Width measurement counter	Displays 2 (A, B) measurement positions (X-, Y-, Z-axes).												
Rotation measurement counter	Displays 3 (A, B, C) measurement positions (X-, Y-axes).												
(2) Manual measurement status display	Displays the manual measurement status. Refer to the "Manual measurement status display" section in "3.3.1 Tool Measurement (M system)" for further details.												
(3) Guide drawing	Displays the measurement image. The contents of the guide drawing differ depending on the measurement pattern (surface measurement, hole measurement, width measurement, or rotation measurement).												
(4) Sensor length and diameter	<p>Sensor length : Displays the length to the tip of the touch sensor. ("#8701 Tool Length" setting value) Sensor diameter : Displays the diameter of the ball at the tip of the touch sensor. ("#8702 Tool Dia" setting value)</p>												
(5) Coordinate system offset	Displays the currently selected coordinate system offset.												
(6) Measurement position counter	<p>Displays the measurement position for all axes. X axis : X axis machine position + sensor radius +center compensation (Horizontal) Y axis : Y axis machine position + sensor radius +center compensation (Vertical) Z axis : Z axis machine position - sensor length 4th axis and over : Respective machine position</p> <table style="margin-left: 40px;"> <tr> <td style="border-left: 1px solid black; border-right: 1px solid black; padding: 0 10px;">Sensor length</td> <td style="padding: 0 10px;">:</td> <td>"#8701 Tool Length"</td> </tr> <tr> <td style="border-left: 1px solid black; border-right: 1px solid black; padding: 0 10px;">Sensor radius</td> <td style="padding: 0 10px;">:</td> <td>"#8702 Tool Dia"/2</td> </tr> <tr> <td style="border-left: 1px solid black; border-right: 1px solid black; padding: 0 10px;">Center compensation (H)</td> <td style="padding: 0 10px;">:</td> <td>"#8703 OFFSET X"</td> </tr> <tr> <td style="border-left: 1px solid black; border-right: 1px solid black; padding: 0 10px;">Center compensation (V)</td> <td style="padding: 0 10px;">:</td> <td>"#8704 OFFSET Y"</td> </tr> </table>	Sensor length	:	"#8701 Tool Length"	Sensor radius	:	"#8702 Tool Dia"/2	Center compensation (H)	:	"#8703 OFFSET X"	Center compensation (V)	:	"#8704 OFFSET Y"
Sensor length	:	"#8701 Tool Length"											
Sensor radius	:	"#8702 Tool Dia"/2											
Center compensation (H)	:	"#8703 OFFSET X"											
Center compensation (V)	:	"#8704 OFFSET Y"											
(7) Center shift amount	Displays the shift amount of the coordinate rotation center.												
(8) Coordinate rotation center	Displays the coordinates that are the center during coordinate rotation. This is the setting value of "#8623 Coord rot centr (H)" and "#8624 Coord rot centr (V)".												
(9) Coordinate rotation angle	Displays the rotation angle during coordinate rotation. This is the setting value of "#8627 Coord rot angle".												

Menus

Menu	Details	Type	Reference
Coord write	The results calculated from the measurement counter are set in the displayed workpiece coordinate system offset. Only the axis for which a value is displayed in the measurement counter can be set. (Note 1) If the workpiece coordinate system to be set is G54 to G59, the local coordinate system offset value will be zero. (Only for the measured axis.) (Note 2) When the parameter "#8924 MEAS. CONFIRM MSG" is "1", this menu is highlighted and the message "OK? (Y/N)" appears. The measurement result is written as tool compensation amount by pressing Y or INPUT key. Even if other key is pressed, the result is not written. When this menu is pressed again while displaying the message, the highlight is released and the operation message is erased.	C	
Coord G54-G59	This selects the displayed workpiece coordinate system from the sub-menu (G54 - G59). Select the workpiece coordinate system to display the selected workpiece coordinate system offset in the coordinate system offset section. (Perform the offset amount setting at the menu <input type="text" value="Coord write"/> key.)	C	
Coord G54.1 P	Input the P number to display the selected extension workpiece coordinate system (G54.1 Pn) offset in the coordinate system offset section. (Perform the offset amount setting at the menu <input type="text" value="Coord write"/> key.) This menu does not display if the extension workpiece coordinate system offset option is disabled.	A	
Coord EXT	This displays the external workpiece coordinate system offset in the coordinate system offset section. (Perform this setting at the menu <input type="text" value="Coord write"/> key.)	C	
SkipPos take in	This creates a false signal when performing simple measurement (measurement without using the touch sensor) and reads the skip position. Press this menu to display the measurement position measured from the machine position of the axis moved last (axis 1 or 2) at the measurement counter. The skip position cannot be read for an axis of auxiliary axis state.	B	
Face meas	This enables surface measurement. Surface measurement is possible when the power is turned ON.	B	3.7.1 Carrying Out Surface Measurement
Hole meas	This enables hole measurement. (The cursor moves to the measurement counter point A.)	B	3.7.2 Carrying Out Hole Measurement
Width meas	This enables width measurement. (The cursor moves to the measurement counter point A.)	B	3.7.3 Carrying Out Width Measurement
Rotate meas	This enables rotation measurement. (The cursor moves to the measurement counter point A.)	B	3.7.4 Carrying Out Rotation Measurement
Next axis	This changes the axes displayed at the coordinate system offset and measurement position counter from axis 1 - 5, and axis 6 and over. (Note) This displays when the number of enabled axes is 6 or over.	C	
Center shift	This sets the center shift amount. (The cursor moves to the 1st axis of "center shift (S)" when this is selected.)	A	3.7.4 Carrying Out Rotation Measurement
Rotate center	This sets the coordinate rotation center. (The cursor moves to the 1st axis of "coordinate rotation center" when this is selected.)	A	
Rotate angle	This sets the coordinate rotation angle. (The cursor moves to the "coordinate rotation angle" when this is selected.)	A	

Details of the "Measurement counter" and "Guide drawing" display areas

Menu	Measurement counter	Guide drawing												
<p>Face meas</p>	<p>Face meas</p> <table border="1"> <tr> <td>X</td> <td>-1566.122</td> </tr> <tr> <td>Y</td> <td>-433.368</td> </tr> <tr> <td>Z</td> <td></td> </tr> </table>	X	-1566.122	Y	-433.368	Z								
X	-1566.122													
Y	-433.368													
Z														
<p>Hole meas</p>	<p>Hole meas</p> <table border="1"> <tr> <td>A:X</td> <td>-1566.122</td> </tr> <tr> <td>Y</td> <td>-433.368</td> </tr> <tr> <td>B:X</td> <td></td> </tr> <tr> <td>Y</td> <td></td> </tr> <tr> <td>C:X</td> <td></td> </tr> <tr> <td>Y</td> <td></td> </tr> </table>	A:X	-1566.122	Y	-433.368	B:X		Y		C:X		Y		
A:X	-1566.122													
Y	-433.368													
B:X														
Y														
C:X														
Y														
<p>Width meas</p>	<p>Width meas</p> <table border="1"> <tr> <td>A:X</td> <td>-1566.122</td> </tr> <tr> <td>Y</td> <td>-433.368</td> </tr> <tr> <td>Z</td> <td></td> </tr> <tr> <td>B:X</td> <td></td> </tr> <tr> <td>Y</td> <td></td> </tr> <tr> <td>Z</td> <td></td> </tr> </table>	A:X	-1566.122	Y	-433.368	Z		B:X		Y		Z		
A:X	-1566.122													
Y	-433.368													
Z														
B:X														
Y														
Z														
<p>Rotate meas</p>	<p>Rotate meas</p> <table border="1"> <tr> <td>A:X1</td> <td>-1566.122</td> </tr> <tr> <td>Y1</td> <td>-433.368</td> </tr> <tr> <td>B:X1</td> <td></td> </tr> <tr> <td>Y1</td> <td></td> </tr> <tr> <td>C:X1</td> <td></td> </tr> <tr> <td>Y1</td> <td></td> </tr> </table>	A:X1	-1566.122	Y1	-433.368	B:X1		Y1		C:X1		Y1		
A:X1	-1566.122													
Y1	-433.368													
B:X1														
Y1														
C:X1														
Y1														

3.7.1.1 Carrying Out Surface Measurement

Outline

When carrying out surface measurement, the position of each axis is measured and the measurement results are set in the workpiece coordinates system offset. The measurement position displays at the measurement counter.

<Measurement using the touch sensor>

Measurement counter X = X axis skip position + sensor radius **(Note)**

Measurement counter Y = Y axis skip position + sensor radius **(Note)**

Measurement counter Z = Z axis skip position - sensor length

(Note) The sign (+ and -) changes depending on the direction of the axis moved last.

Item	Reference
Sensor radius	"#8702 Tool Dia"/2
Sensor length	"#8701 Tool Length "

<Simple measurement (measurement without using touch sensor)>

Measurement counter X : X axis measurement position

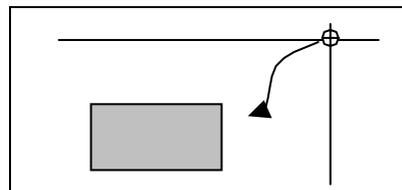
Measurement counter Y : Y axis measurement position

Measurement counter Z : Z axis measurement position

Operation method

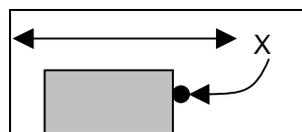
- (1) Press the **Face meas** menu. → The menu is highlighted.
- (2) Perform the reference position return command, etc. to position the tool at the basic point.
- (3) Turn ON the measurement switch on the machine operation board. → The message "On meas" appears.

- (4) Move the tool near the workpiece using manual feed and manual handle feed.



<With the touch sensor>

- (5) Move the tool in the X-direction until the sensor contacts the workpiece. → Upon contact, the axis automatically contacts the workpiece again. The measurement position measured from the skip position displays in the measurement counter X axis section.



- <Without the touch sensor (simple measurement)>
- (5) Move the tool in the X-direction to the arbitrary position and press the menu SkipPos take in . ➔ The skip position is calculated and the result displays in the measurement counter X axis section.
 - (6) Select the workpiece coordinate system offset in which the measurement data will be set.
(Example) Press the menus Coord G54-G59 G55 to select G55.
 - (7) Press the Coord write menu. ➔ The value achieved by adding or subtracting the external workpiece offset to or from the X axis measurement position based on the "#8709 Ext work sign rvs" is set in workpiece coordinate system selected at (6).
The measurement counter displays blank.
 - (8) Carry out the operations in steps (4) to (7) in the same way for the Y and Z axes.
 - (9) Return the tool to the reference position, and turn the measurement switch OFF. ➔ The message "On meas" disappears.

3.7.1.2 Carrying Out Hole Measurement

Outline

When carrying out hole measurement, three positions (A, B, and C) are measured, and the hole center position calculated from the three measured positions is set in the workpiece coordinate system offset. The measurement position displays at the measurement counter.

<Measurement using the touch sensor>

Measurement counter X = X axis skip position
Measurement counter Y = Y axis skip position

<Simple measurement (measurement without using touch sensor)>

Measurement counter X = X axis machine position + center compensation (H) + skip past amount (Horizontal axis) **(Note)**
Measurement counter Y = Y axis machine position + center compensation (V) + skip past amount (Vertical axis) **(Note)**

(Note) The skip past amount is added for only the axis that moved last.
The sign (+ or -) of the past amount depends on the movement direction of the axis.

Item	Reference
Center compensation (H)	"#8703 OFFSET X"
Center compensation (V)	"#8704 OFFSET Y"
Skip past amount (horizontal axis)	"#8707 Skip past amout (H)"
Skip past amount (vertical axis)	"#8708 Skip past amout (V)"

Operation method

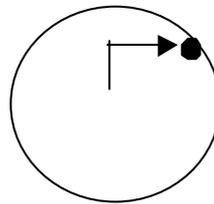
- (1) Press the menu. → The menu is highlighted and the cursor displays at point A.
Use the , keys to move to points B and C.

- (2) Perform the reference position return command, etc. to position the tool at the basic point.

- (3) Turn ON the measurement switch on the machine operation board. → The message "On meas" appears.

- (4) Move the tool into the hole using manual feed and manual handle feed.

- <With the touch sensor>
(5) Position the tool against the inner wall of the hole. Contact is performed by moving a single axis. →



Upon contact, the axis automatically contacts the hole wall again. The skip position displays in the measurement counter point A, X and Y axis section. The cursor moves to point B. For point C, the cursor moves to point A.

- <Without the touch sensor (simple measurement)>
(5) Move the tool in the X-direction to the arbitrary position and press the menu . → The skip position is calculated and the result displays in the measurement counter X axis section.
The cursor moves to point B. For point C, the cursor moves to point A.

- (6) Measure points B and C in the same way.

- (7) Select the workpiece coordinate system offset in which the measurement data will be set.
(Example) Press the menus to select G55.

- (8) Press the menu. → The hole center position is measured, and based on the "#8709 Ext work sign rvs", value achieved by adding or subtracting the external workpiece to or from the measurement result is set in workpiece coordinate system selected at (7). The measurement counter displays blank.

- (9) Return the tool to the reference position, and turn the measurement switch OFF. → The message "On meas" disappears.

3.7.1.3 Carrying Out Width Measurement

Outline

When carrying out width measurement, two positions (A, and B) are measured, and the width center position calculated from the two measured positions is set in the workpiece coordinate system offset. The measurement position displays at the measurement counter.

<Measurement using the touch sensor>

Measurement counter X = X axis skip position

Measurement counter Y = Y axis skip position

Measurement counter Z = Z axis skip position - sensor length

<Simple measurement (measurement without using touch sensor)>

Measurement counter X = X axis machine position + center compensation (H) + skip past amount
(Horizontal axis) **(Note)**

Measurement counter Y = Y axis machine position + center compensation (V) + skip past amount
(Vertical axis) **(Note)**

Measurement counter Z = Z axis machine position - sensor length

(Note) The skip past amount is added for only the axis that moved last.

The sign (+ or -) of the past amount depends on the movement direction of the axis.

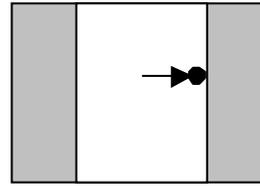
Item	Reference
Sensor length	"#8701 Tool Length"
Center compensation (H)	"#8703 OFFSET X"
Center compensation (V)	"#8704 OFFSET Y"
Skip past amount (horizontal axis)	"#8707 Skip past amount (H)"
Skip past amount (vertical axis)	"#8708 Skip past amount (V)"

Operation method

- (1) Press the Width meas menu.  The menu is highlighted and the cursor displays at point A.
Use the ,  keys to move to points A and B.
- (2) Perform the reference position return command, etc. to position the tool at the basic point.
- (3) Turn ON the measurement switch on the machine operation board.  The message "On meas" appears.
- (4) Move the tool to the center of the groove using manual feed or manual handle feed.

<With the touch sensor>

- (5) Position the tool against the inner wall of the groove. Contact is performed by moving a single axis.



Upon contact, the axis automatically contacts the groove wall again. The skip position displays in the measurement counter point A.
The cursor moves to point B. For point B, the cursor moves to point A.

<Without the touch sensor (simple measurement)>

- (5) Move the tool to the arbitrary position and press the menu `SkipPos take in`.



The skip position is calculated and the results displays in the measurement counter X axis and Y axis section.
The cursor moves to point B. For point B, the cursor moves to point A.

- (6) Similarly, position the tool against the opposite side.



Upon contact, the axis automatically contacts the groove wall again. The skip position displays in the measurement counter point B.

- (7) Select the workpiece coordinate system offset in which the measurement data will be set.

(Example) Press the menus `Coord G54-G59` `G55` to select G55.

- (8) Press the `Coord write` menu.



The width center position is measured, and based on the "#8709 Ext work sign rvs", value achieved by adding or subtracting the external workpiece offset to or from the measurement result is set in workpiece coordinate system offset G55 (coordinate system selected at (7)). The measurement counter displays blank.

- (9) Carry out the operations in steps (4) to (8) in the same way for the Y and Z axes.

- (10) Return the tool to the reference position, and turn the measurement switch OFF.



The message "On mea" disappears.

3.7.1.4 Carrying Out Rotation Measurement

Outline

When carrying out rotation measurement, the offset (rotation center and rotation angle) of the rotary coordinate system is measured, and the results are set to the workpiece coordinate system offset (rotation center) and the parameters "#8623 Coord rot centr(H)", "#8624 Coord rot centr(V)", and "#8627 Coord rot angle".

<Measurement using touch sensor>

Measurement counter X = X axis skip position (Machine position)

Measurement counter Y = Y axis skip position (Machine position)

<Simple measurement (measurement without using touch sensor)>

Measurement counter X = X axis machine position + center compensation (H) + skip past amount (Horizontal axis) **(Note)**

Measurement counter Y = Y axis machine position + center compensation (V) + skip past amount (Vertical axis) **(Note)**

(Note) The skip past amount is added for only the axis that moved last.

The sign (+ or -) of the skip past amount depends on the movement direction of the axis.

Item	Reference
Center compensation (H)	"#8703 OFFSET X"
Center compensation (V)	"#8704 OFFSET Y"
Skip past amount (horizontal axis)	"#8707 Skip past amout (H)"
Skip past amount (vertical axis)	"#8708 Skip past amout (V)"

Operation method (Setting the center shift amount)

- (1) Press the Rotate meas menu.

The pop-up window for the rotation measurement appears.
The measurement counter becomes active and the cursor appears at the 1st axis.
- (2) Press the Center shift menu.

The menu is highlighted.
"C shift (S)" column becomes active and the cursor appears at the 1st axis.
The measurement counter active display returns to normal.
- (3) Input the center shift amount (Si).
10.000 INPUT

The value is input and the cursor moves to the 2nd axis.
- (4) Input the center shift amount (Sj).
-5.000 INPUT

The value is input, however, the cursor does not move.
- (5) Press the Center shift menu.

The menu highlight returns to normal.
The measurement counter becomes active and the cursor appears at the 1st axis.
"C shift (S)" column active display returns to normal.

Operation method (Setting the center and angle of the coordinate rotation)

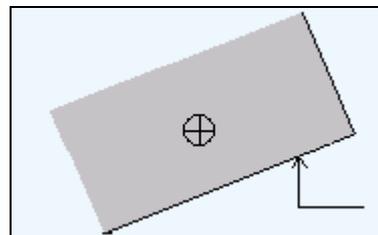
The setting of the parameters "#8623 Coord rot centr(H)", "#8624 Coord rot centr(V)", and "#8627 Coord rot angle" can be changed.

- (1) Press the menu.  The pop-up window for the rotation measurement appears.
The measurement counter becomes active and the cursor appears at the 1st axis.
- (2) Press the menu.  The menu is highlighted.
"Coord center" column becomes active and the cursor appears at the 1st axis.
The measurement counter active display returns to normal.
- (3) Input data in the 1st axis of "Coord center" column.  The value is input and the cursor moves to the 2nd axis.
- (4) Input data in the 2nd axis of "Coord center" column.  The value is input, however, the cursor does not move.
- (5) Press the menu.  The menu highlight returns to normal.
The measurement counter becomes active and the cursor appears at the 1st axis.
"Coord center" column active display returns to normal.

(Note) Set the angle of the coordinate rotation in the same manner.

Operation method (Carrying out the rotation measurement using touch sensor to set the measurement results to the workpiece coordinate offset)

- (1) Press the **Rotate meas** menu. → The menu is highlighted and the cursor appears at point A.
The cursor can be moved to A, B, or C, using **↑** and **↓** keys.
- (2) Set the parameter "#8116 Coord rot para invd" to "1" (invalid).
- (3) Set the following parameters to "0". → The value is input, however, the cursor does not move.
#8623 Coord rot centr(H)
#8624 Coord rot centr(V)
#8627 Coord rot angle
(Note) These parameter should be set to "0" to set the measured value to them.
If the value other than "0" has been set, an error occurs when writing to the coordinate system.
- (4) Turn the manual absolute switch ON. → The value is input and the cursor moves to the 2nd axis.
- (5) Perform the reference position return command, etc. to position the tool at the basic point.
- (6) Turn ON the measurement switch on the machine operation board. → The message "On meas" appears.
- (7) Referring to "Setting the center shift amount", set the center shift amount. → The menu highlight returns to normal.
This setting is not necessary if the coordinate rotation center is served as the workpiece coordinate zero point.
The measurement counter becomes active and the cursor appears at the first axis.
"Coord center" column active display returns to normal.
- <Measurement using touch sensor>
- (8) Make the spindle contact the workpiece using manual feed or manual handle feed. →



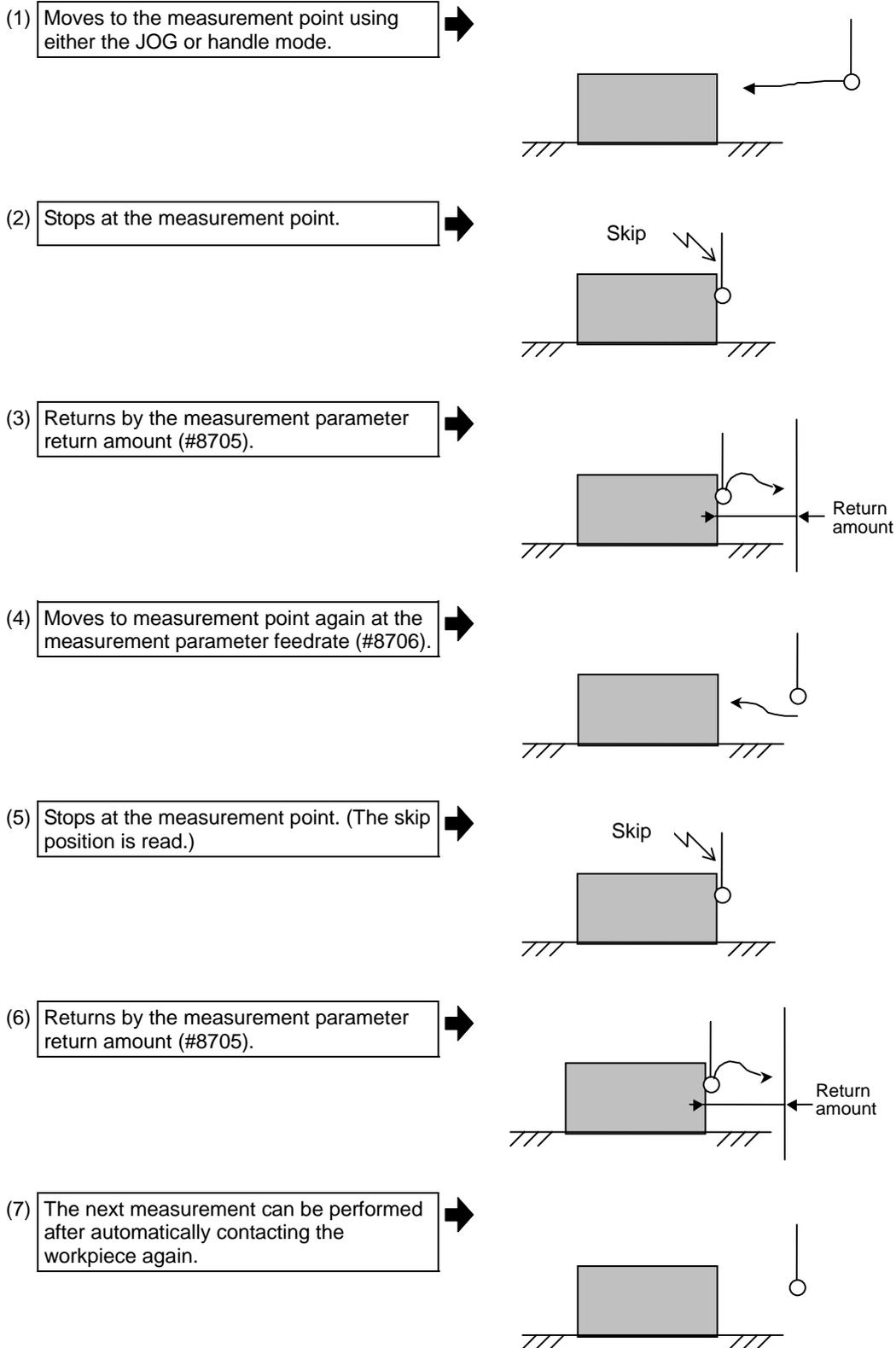
Upon contact, the skip position displays in the measurement counter point A.
The cursor moves to point B. For point C, the cursor moves to point A.

<Simple measurement (without using touch sensor)>

- (8) Move the spindle to the arbitrary position and press the `SkipPos take in` menu. → The skip position is calculated and the results appear in the measurement counter point A.
The cursor moves to point B. For point C, the cursor moves to point A.
- (9) Measure points B and C in the same way.
- (10) Select the workpiece coordinate system offset in which the measurement data will be set.
(Example) To select G55, press the `Coord G54-G59` `G55` menus.
- (11) Press the `Coord write` menu. → The hole center position is measured, and based on the "#8709 Ext work sign rvs", value achieved by adding or subtracting the external workpiece to or from the measurement result is set in workpiece coordinate system selected at (10). The measurement counter displays blank.
- (12) Return the tool to the reference position, and turn the measurement switch OFF. → The message "On meas" disappears.

3.7.1.5 Performing Automatic Recontact When Contacting the Workpiece

When performing workpiece position measurement in either the JOG or handle mode, the following operations (automatic recontact) are performed after contacting the workpiece.



The return speed for (3) and (6) is 40 times the measurement parameter feedrate (#8706). However, if the return speed is faster than rapid feedrate (override 100%), the rapid feedrate will become the return speed.

3.7.2 Workpiece Measurement (L system)

The external workpiece coordinate offset data for the Z axis can be set by cutting the workpiece face by means of manual operations and inputting the workpiece measurement signal.

By pressing the menu key, data can be set in the Z axis of an arbitrary coordinate offset.

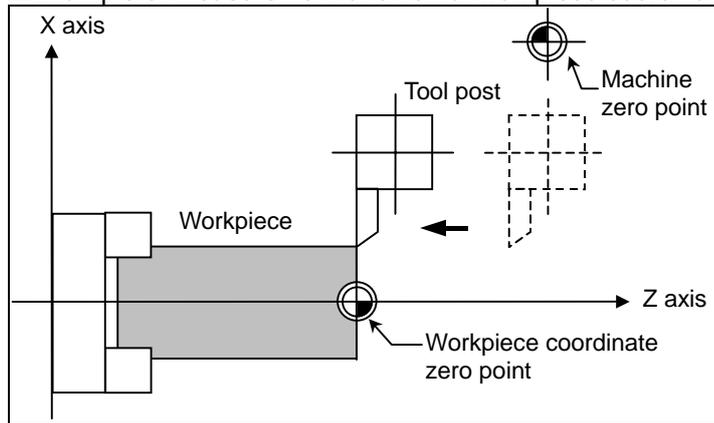
Coordinate offset setting for the Z axis (2nd axis) of the part system 1 to 4 is possible.

Note that the workpiece coordinate offsets for multiple part systems cannot be measured at the same time.

For signals and R registers, refer to the instruction manual issued by the machine tool builder.

(Note) Measurement is disabled for the part system with one-axis structure.

Example of measurement of external workpiece coordinate offset data for Z axis



Manual numerical value command can be executed when manual numerical value command window is displayed by inputting an address key such as MST, etc.

Display items

Display item	Details
(1) External workpiece coordinate offset	This displays the external workpiece coordinate offset. When the workpiece measurement signal is input, the measurement result is set to 2nd axis of the external workpiece coordinate. (Note) The 5th axis and following are not displayed.
(2) Tool offset num	This displays the compensation No. of tool length data used by automatic calculation. (Note 1) If an illegal value is set, the value is blank. ("0" is displayed.) (Note 2) The No. is displayed by BCD code.
(3) Wear offset num	This displays the compensation No. of tool nose wear data used by automatic calculation. (Note 1) If an illegal value is set or tool nose wear data is not used for measurement (" #1226 aux10/bit0" is 1), the value is blank. ("0" is displayed.) (Note 2) The No. is displayed by BCD code.
(4) Coordinate system offset	This displays the coordinate system offset in which the measurement result is written when "MeasVal take in" menu is pressed. G54 is displayed in default. When "MeasVal take in" menu is pressed, the measurement result is set to 2nd axis of the external workpiece coordinate. (Note 1) If an illegal value is set, the value is blank. ("0" is displayed.) (Note 2) The display of Coordinate system offset area is switched by page changeover keys. Refer to the section "3.6.4 Changing the Coordinate System Display".
(5) On mea	When the measurement switch is ON and the operation mode is manual mode ([Handle], [Jog], [Rapid Traverse], or [Step]), "On mea" is displayed, and the workpiece can be measured by the workpiece measurement signal.
(6) Guide drawing	This displays the measurement area.

Menus

Menu	Details	Type	Reference
Coord G54-G59	This selects the coordinate system offset which sets the measurement value when "MeasVal take in" is executed from sub-menu (G54 - G59). The workpiece coordinate system offset data is displayed after selected. However, the cursor is not displayed and the data cannot be set.	C	
Coord G54.1 P	This selects the coordinate system offset which sets the measurement value when "MeasVal take in" is executed. Input the P No. to display the selected external workpiece coordinate system (G54.1 Pn) offset in the coordinate system offset area. However, the cursor is not displayed and the data cannot be set. If the option of external workpiece coordinate system offset is invalid, this menu is displayed in gray and cannot be selected.	A	
MeasVal take in	This measures G54 - G59 and G54.1Pn. When this menu is pressed, the workpiece coordinate offset data is calculated from the machine coordinate, the used tool length, the tool nose wear compensation amount and the external workpiece coordinate offset, and the data is stored in Z axis of the selected coordinate system offset.	B	

Operation method (When the workpiece measurement signal is used)

- (1) Execute the reference position return.  The coordinate system is established.
(Note 1) If the workpiece is measured without establishing the reference position return, the operation message "Meas axis not returned to ref. position" appears.
- (2) Set the mode selection switch to the manual mode ([Handle], [Jog], [Rapid Traverse]) or [Step].
- (3) Set "1" to "Tool length measurement 2" signal.
- (4) Select whether to measure the workpiece coordinate on the main spindle side or sub-spindle side by using tool presetter sub-side valid signal.
(Note 1) Hold the state of this signal until the measurement with the selected tool is completed.
- (5) Issue the T command with MDI operation, etc.  The tool is selected.
(Note 1) Set the selected tool compensation No. in R register.
 The set R register differs according to parameter setting and the state of the tool presetter sub-side valid signal.
(Note 2) Set the "tool length/wear data" for the tool being used beforehand.
- (6) Cut the workpiece face slightly to even it.
(Note 1) Do not move the tool in the detection of Z axis after cutting the workpiece face.
(Note 2) If the workpiece does not need to be cut, carry out positioning to the measurement position.
- (7) Input the workpiece measurement signal.  The Z axis external workpiece coordinate offset data is automatically calculated from the tool compensation data of the used tool and the machine value at the point this signal is input. The results are set as the data.
- (8) Turn "Tool length measurement 2" signal OFF.  Measurement of the external workpiece coordinate offset is completed.

Operation method (When the **MeasVal take in** menu is used)

- (1) Execute the reference position return. → The coordinate system is established.
(Note 1) If the workpiece is measured without establishing the reference position return, the operation message "Meas axis not returned to ref. position" appears.
- (2) Set the mode selection switch to the manual mode ([Handle], [Jog], [Rapid Traverse]) or [Step].
- (3) Select an arbitrary coordinate system offset. → The designated coordinate system offset is selected.
(Note 1) When switching to an arbitrary coordinate system offset display, press one of these; "Coord G54 - G59" "Coord G54.1P" menu or page key.
- (4) Select whether to measure the workpiece coordinate on the main spindle side or sub-spindle side by using tool presetter sub-side valid signal.
(Note 1) Hold the state of this signal until the measurement with the selected tool is completed.
- (5) Issue the T command by MDI operation, etc. → The tool is selected.
(Note 1) Set the selected tool compensation No. in R register.
 The set R register differs according to parameter setting and the state of the tool presetter sub-side valid signal.
(Note 2) Set the "tool length/wear data" for the tool being used beforehand.
- (6) Cut the workpiece face slightly to even it.
(Note 1) Do not move the tool in the detection of Z axis after cutting the workpiece face.
(Note 2) If the workpiece does not need to be cut, carry out positioning to the measurement position.
- (7) Press the menu **MeasVal take in**. → The workpiece measurement is executed.
 The operation message "OK? (Y/N)" appears, so press the **Y** or **INPUT**.
 The Z axis external workpiece coordinate offset data is automatically calculated from the tool compensation data of the used tool and the machine value at the point this signal is input. The results are set as the data.

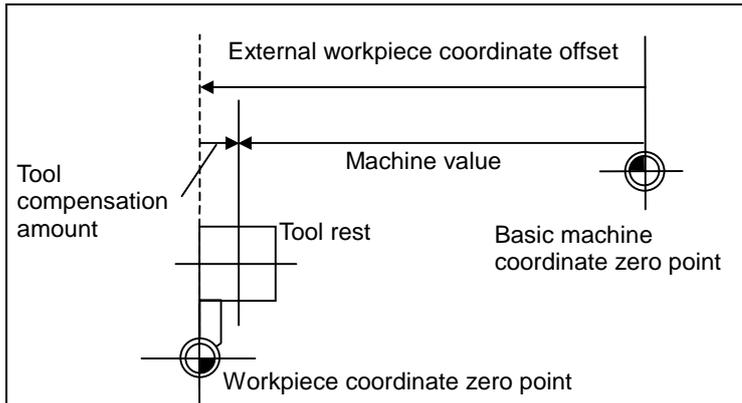
Details of automatic calculation expression (When the workpiece measurement signal is used)

The external workpiece coordinate offset data is automatically calculated with the following expression.

$$\text{External workpiece coordinate offset} = \text{Machine coordinate value} - \text{Tool compensation data}$$

The tool compensation data used for the measurement is selected with the base specification parameter "#1226 aux10 bit0".

#1226 aux10 bit0	Tool compensation data
0	Tool length data + nose wear data
1	Tool length data



Details of automatic calculation expression (When the MeasVal take in menu is used)

The workpiece coordinate offset data is automatically calculated with the following expression.

The calculation expression changes according to basic specification parameter "#8709".

When "#8709" is "0":

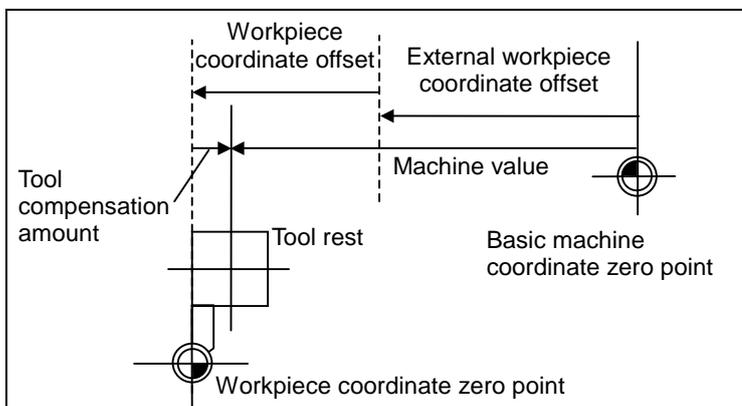
$$\text{Workpiece coordinate offset} = \text{Machine coordinate value} - \text{External workpiece coordinate offset} - \text{Tool compensation data}$$

When "#8709" is "1":

$$\text{Workpiece coordinate offset} = \text{Machine coordinate value} + \text{External workpiece coordinate offset} - \text{Tool compensation data}$$

The tool compensation data used for the measurement is selected with the base specification parameter "#1226 aux10 bit0".

#1226 aux10 bit0	Tool compensation data
0	Tool length data + nose wear data
1	Tool length data



The selected tool compensation No.

For the compensation No. of the tool length and tool nose wear data used by automatic calculation, the No. set in R register in the following table is used.

#1098 TIno.	#1130 set_t	#1218 aux02 bit4	Tool length compensation No.		Tool nose wear compensation No.	
			Main side	Sub-side	Main side	Sub-side
0	0/1	0/1	R2600, R2601	R2604, R2605	R2600, R2601	R2604, R2605
1	0	0	R536, R537			
		1	R2602, R2603	R2606, R2607		
	1	0/1				

- (Note 1)** When the compensation No. is 0, the compensation amount will be calculated as "0".
- (Note 2)** If the compensation No. exceeds the number of specified offset sets, the operation message "Offset No. not found" appears.
- (Note 3)** If the input data exceeds the setting range, the operation message "Data range error" appears.
- (Note 4)** If the data protect key 1 is valid, the operation message "Data protect" appears.
- (Note 5)** Whether to use the main side or sub-side is selected with the tool presetter sub-side valid signal. (OFF: Main side, ON: Sub-side)
- (Note 6)** Set the compensation No. in R register by the BCD code.

3.8 User Parameters



The parameters contain the user and machine parameters. This chapter explains the details and operations for the user parameters. Refer to "Appendix 9. User Parameter List" for details of each parameter.

<High-accuracy parameters>: Part system based

Relat posn	No.	Name	Data
X	1149	cireft	0
YM	1205	G0bdcc	1
YS	1206	G1bF	5000
Z	1207	G1btL	50
	1209	cirdcc	99999
Mach posn			
X	1568	SfiltG1	30
YM	1569	SfiltG0	45
YS	1570	Sfilt2	25
Z	1571	SSSdis	0
S	8019	R COMP	0
M		Theor R decrease	0.061
	8020	DCC. ANGLE	89

<High-accuracy parameters>: Axis based

Relat posn	No.	Name	X	YM	YS
X	2001	rapid	48000	48000	36000
YM	2002	clamp	36000	36000	36000
YS	2010	fwd_g	45	45	45
Z	2068	G0fwdg	0	0	0
	2096	crncsp	0	0	0
Mach posn					
X	2109	Rapid(H-precision)	200000	200000	200000
YM	2110	Clamp(H-precision)	200000	200000	200000
YS					
Z					

Display items

Display item	Details
(1) Parameter display area	Press the parameter selection menu <input type="button" value="Process param"/> or <input type="button" value="Hi-prec param"/> keys etc. to change the parameter display contents.
(2) Input section	The set key displays. Press the <input type="button" value="INPUT"/> key to set the display data in the parameter where the cursor is located.

Menus

Menu	Details	Type	Reference
<input type="button" value="Process param"/>	This changes the display of each parameter type. (Note) The barrier data only displays for the L system.	B	3.8.2 Setting the Parameters
<input type="button" value="Fixed cycle"/>			
<input type="button" value="Ctrl param 1"/>			
<input type="button" value="Ctrl param 2"/>			
<input type="button" value="I/O param"/>			
<input type="button" value="Ethernet param"/>			
<input type="button" value="Link param"/>			
<input type="button" value="Subpro stor"/>			
<input type="button" value="Axis param"/>			
<input type="button" value="Barrier data"/>			
<input type="button" value="Hi-prec param"/>			
<input type="button" value="Hi-prec axis"/>			
<input type="button" value="Operate param"/>			
<input type="button" value="Param No."/>	This selects an arbitrary parameter number. Set the parameter number and press the <input type="button" value="INPUT"/> key to move that number to the top and display the parameter. The cursor also moves to that number.	A	3.8.1 Selecting the Parameter Number
<input type="button" value="Area copy"/>	This copies the parameter setting values in the designated range. The range is designated with numbers.	A	3.8.3 Copying/Pasting Parameters
<input type="button" value="Area paste"/>	This pastes the range of parameters designated in area copy. They are pasted in a parameter corresponding to the axis or part system where the cursor is. Once copied, a parameter can be pasted any number of times until a new parameter is copied.	A	
<input type="button" value="Next axis"/>	This can be selected if the total number of part system display axes is 5 or more. This is used at the parameter screen with the array structure for each axis.	C	
<input type="button" value="Close"/>	This closes the pop-up window and quits this function.	C	

3.8.1 Selecting the Parameter Number

Press the menu key to select an arbitrary parameter number.

3.8.2 Setting the Parameters

The method of setting the parameters is explained. Refer to "3.8.8 User Parameters" for details of each parameter setting range.

Operation method (Normal setting: Set "100" for X1 axis of "#8204 OT-")

- (1) menu."/> ➔ The axis parameter displays.
- (2) , , , and keys to move the cursor to the setting position."/> ➔ The cursor moves to the position where the data is to be set (X1 axis).
- (3) "/> ➔ The set value displays and the cursor moves one position downwards.

Operation method (Batch data setting: Set "200" for Y1 axis and "250" for Z1 axis of "#8204 OT-")

- (1) menu."/> ➔ The axis parameter displays.
- (2) , keys to move the cursor to the setting position."/> ➔ The cursor moves to the position where the data is to be set.
- (3) "/> ➔ The set value displays and the cursor moves one position downwards.

(Note 1) When a parameter is changed and displays at the top of the screen, reboot the machine to enable that parameter change.

(Note 2) When the key is pressed without inputting a value, the parameter setting value is not changed, and the cursor moves.

(Note 3) Parameters that can be set simultaneously must display within the currently displayed three columns.

(Note 4) If parameter values are simultaneously input in multiple columns, settings are made from the currently displayed column on the left side, no matter which column the cursor is positioned.

3.8.3 Copying/Pasting Parameters

Parameters can be copied and pasted in a parameter with the same number for another axis or part system.

Operation method (Specifying and copying the parameter where the cursor is located)

- (1) Press the menu.  The menu is highlighted.
- (2) Move the cursor to the position of the parameter to be copied and press the key.  The copy position displays in the selected area color. The menu highlight returns to normal.

Operation method (Copying parameter numbers in a specific range)

- (1) Move the cursor to the display area of the axis or part system to be copied.
- (2) Press the menu.  The menu is highlighted.
- (3) Designate the copy range.
Format: First No./Last No.
8701/8705  The range to be copied displays in the selected area color. The menu highlight returns to normal.

An "E" can be used if the last number to be copied is the last number in the currently displayed parameter type.

(Ex.) 8701/E

No.	Name	Data
	<TLM>	
8701	Tool Length	100.000
8702	Tool Dia	0.000
8703	OFFSET X	5.000
8704	OFFSET Y	10.000
8705	RETURN	5.000
8706	FEED	100
8707	Skip past amount (H)	0.000

(Note 1) The selected area color of the copied area returns to normal when the copied data is pasted.

However, the copied range can still be pasted until the parameter type display is changed.

(Note 2) Only the currently displayed parameter type can be copied.

Operation method (Pasting the copied parameters)

- (1) Move the cursor to the axis or part system display area where the data will be pasted.
- (2) Press the menu.  The menu is highlighted and a confirmation message displays.
- (3) Press the or key.  The copied data is written to the parameter where the cursor is located.
The copied data is not written in when the key is pressed. The menu highlighted returns to normal.

(Note) If the relevant parameter is changed after specifying the range to be copied, the value after the change is pasted.

3.8.4 Parameter Configuration

The parameter number range and permissible operations for all parameter types are shown below.

○: Can be used, △: Operable under certain conditions (Refer to the section below for conditions.)

Parameter type	Next axis menu	System change Key	Area copy menu	Area paste menu
Process parameter	-	△ (Note 3)	△ (Note 3)	△ (Note 3)
Fixed cycle	-	△ (Note 3)	△ (Note 3)	△ (Note 3)
Control parameter 1	-	-	-	-
Control parameter 2	-	-	-	-
I/O parameter	-	-	-	-
Ethernet parameter	-	-	-	-
Computer link parameter	-	-	-	-
Subprogram storage designation parameter	-	△ (Note 3)	△ (Note 3)	△ (Note 3)
Axis parameter	△ (Note 1)	-	△ (Note 2)	△ (Note 2)
Barrier data (Note 4)	-	△ (Note 3)	○	○
High precision parameter	-	△ (Note 3)	△ (Note 3)	△ (Note 3)
High precision axis parameter	△ (Note 1)	-	△ (Note 2)	△ (Note 2)
Operation parameter	-	-	-	-

(Note 1) This is valid only when the total number of valid NC axes and PLC axes in the entire part system is four or more.

(Note 2) This is valid only when the total number of valid NC axes and PLC axes in the entire part system is two or more.

(Note 3) This is valid only when the number of valid part systems is two or more.

(Note 4) The barrier data appears only for the L system specifications.

3.8.5 Echoback

When the cursor moves to a parameter with 14 or more characters, all of the set value is echoed back to the input area. The value of such parameter is displayed with "..." displays in the data field.

No.	Name	Data
9701	IP addr auto set	1
9702	IP address	100.97.12.1
9703	Subnet mask	255.255.255.0
9704	Gateway	100.97.12.254
9705	Timeout	0
9706	Host No.	1
9711	Host1 host name	MELCO
9712	Host1 user name	USER1
9713	Host1 password	*****
9714	Host1 directory	/MITSUBISHI/...
9715	Host1 host type	1
9716	Host1 Wrđ pos:name	9
9717	Host1 Wrđ pos:size	5
9718	Host1 Wrđ pos:Dir	1

/MITSUBISHI/MOLD1/EM0001

3.8.6 Ethernet Parameter Password

If the Ethernet parameter "Host 1 password" to "Host 4 password" is set, "*****" displays regardless of the password characters set in that data field.

3.8.7 Machine Parameter Password Setting Method

The "High-accuracy parameters" and "High-accuracy axis parameters" contain machine parameters other than the user parameter. It is possible to reference these machine parameters, however, if a machine tool builder password is not set, it is not possible to set any data.

Move the cursor to the machine parameters to display a "Input a password." message, and the system waits for the password to be input. Input the machine tool builder password as required. Once input, the password remains valid until the power is turned OFF.

Operation method (In the case where the password has not been input even once after turning ON the power.)

- (1) Display the High-accuracy Parameters screen.

When the ,  keys are used to move the cursor to a position other than the machine parameters, the awaiting password status is cleared along with the message.



When the cursor is positioned at the machine parameters, the "Input a password." message displays. (System awaits password)

No.	Name	Data
1149	ci reft	0
1205	G0bdcc	0
1206	G1bF	30000
1207	G1btL	100
1209	ci rdcc	1
1568	SfiltG1	0
1569	SfiltG0	0
1570	Sfilt2	0
1571	SSSdis	0
8019	R COMP	0
	Theor R decrease	2.296

- (2) Input the machine parameter password and press the  key.



"*****" displays at the input area while the password is being input.

Press the  key to clear the awaiting password status, and enable machine parameter settings to be made.

(Note) If the password is input incorrectly, the awaiting password status remains cleared and a "The password is incorrect." message displays.

3.9 MDI Program Editing



Press the main menu [MDI] key to display the contents of the MDI program in a pop-up window. When the parameter “#1144 mdlkof” is “0” (MDI setting impossible), switch to the MDI mode once to press the menu “MDI”.

Refer to "4.2 Program Editing" for further details.



3.11 Manual Numerical Value Commands



This function is used to set and display all the Spindle function (S), Miscellaneous function (M), Tool function (T), and No. 2 miscellaneous function (B) commands.

Input the address for S, M, T, and B to set and display the manual numerical value commands.

Refer to "2.13 Manual Numerical Value Commands" for further details.

SYS1		MDI		SET		Monitr		Setup		Edit		Diagn		Mainte	
MSTB		T-ofs		T-meas		T-reg		T-life		Coord		W-meas			
		Length		L wear		Radius		R wear							
Rela	S	3000	1	0.000	0.000	10.000	0.000								
X1	M	3	2	0.000	0.000	0.000	0.000								
Y1	T	5	3	-500.000	0.000	0.000	0.000								
Z1	B	0	4	0.000	0.000	0.000	0.000								
C1			5	0.000	0.000	0.000	0.000								
Mact			6	0.000	0.000	0.000	0.000								
X1			7	0.000	0.000	0.000	0.000								
Y1			8	0.000	0.000	0.000	0.000								
Z1			9	0.000	0.000	0.000	0.000								
C1			10	0.000	0.000	0.000	0.000								
S			11	0.000	0.000	0.000	0.000								
M			12	0.000	0.000	0.000	0.000								
Spindle		3	13	0.000	0.000	0.000	0.000								
Wait		5	14	0.000	0.000	0.000	0.000								
			15	0.000	0.000	0.000	0.000								
\$1		\$2													
			15:12												
			T-ofs	T-meas	T-reg	T-life	Coord	W-meas	User	MDI	Cnt set	MST			
												Close			

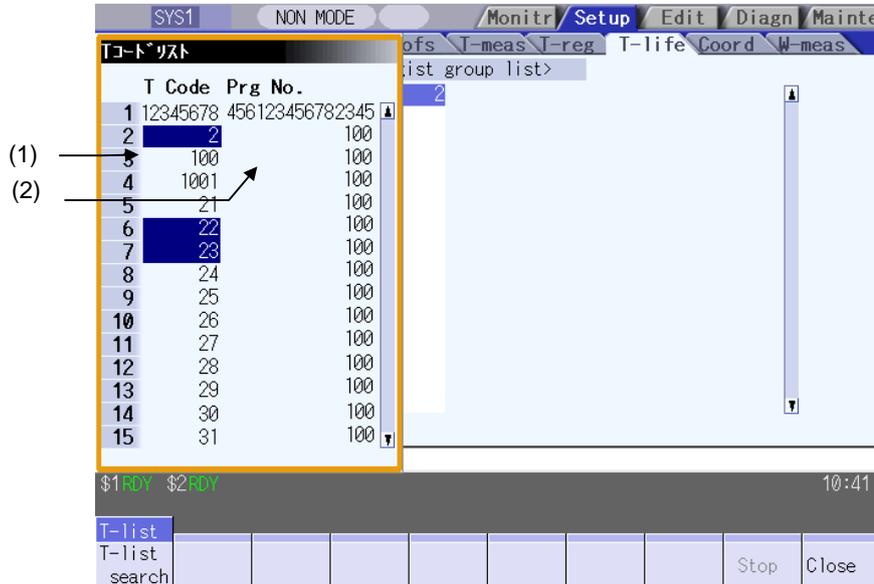
3.12 T Code List



This function searches the T code in designated machining program (including the subprograms) and lists it in the order the T codes are commanded. (Max. 100 codes)

As for the T code of the tool that has not been registered, it is indicated with text color and background color changed.

This window is displayed only when the graphic check option is provided.



Display items

Display item	Details
(1) T code	T codes are listed in the order that the T code is commanded in the designated machining program (including subprogram). As for the T code of the tool that has not been registered, it is indicated with text color and background color changed.
(2) Program No.	This shows designated program No. When the number of characters exceeds 15, the excess is indicated as "**".

Menus (T code list display window)

Menu	Details	Type	Reference
T-list search	This displays T code list search window. This menu is invalid while getting the T codes.	C	
Halt	This interrupts getting the T codes. This menu is valid only while getting the T codes.	C	
Close	This closes the pop-up window and quits this function.	C	

Menus (T code list search window)

Menu	Details	Type	Reference
Memory	This selects the device for searching for the program to get T codes. When a device with directory is selected, the root directory is selected first. (In 70 series, only "Memory" and "Memory card" are displayed.)	C	
HD		C	
Memory card		C	
DS		C	
FD		C	
List update	This updates the list contents. (The latest contents of the currently selected device and directory are listed.)	C	
Sort change	This changes the method that the list is sorted.	C	2.2.3 Changing the Sorting Method
Comment nondisp	This changes whether to show or hide the comment field in the list. When the comment field is hidden, the file name field will be enlarged.	B	2.2.2 Changing Whether to Show or Hide the Comment Field
Retn	This closes the pop-up window and starts getting T codes.	C	

Operation method (Display the T code list)

- (1) Press the menu . → The menu appears.
The T code list created last appears as a pop-up window. The menu becomes invalid.
- (2) Press the menu . → The submenu appears. The list to select a file appears as a pop-up window.
- (3) Select the device.
 → The selected device name and its root directory (HD:/) appears in the directory display area.
- (4) Use the , , , keys to align the cursor with the directory of target machining program, and press . → T code list appears as a pop-up window.
Getting T code and listing starts.
The menu becomes valid.
The is invalid.

T code list cannot be displayed when key is pressed with the following programs selected;

- The program that does not exist.
- The program that is waiting to be restarted.
- The program during automatic operation.
- The program being checked.

The list window will not change.

(Note 1) When switching to the other screen, getting T code is interrupted automatically.

(Note 2) When the menu is pressed, getting T code is interrupted automatically.

(Note 3) T code list for MDI program cannot be displayed.

(Note 4) T code list acquisition is not possible for a program with 33 or more program name characters.

(Note 5) There is a model which cannot get T code list at the manual operation mode. Switch to the automatic operation mode when T code list cannot be gotten.



3.13 Pallet Program Registration [700 series only]

Machining programs can be registered per each pallet of the automatic pallet changer (hereinafter APC) and the indexing plane.

The Pallet Program Registration screens include "Standard Pallet Registration screen" and "Pallet 4 Page Registration screen", and those two screens can be switched by the parameters.

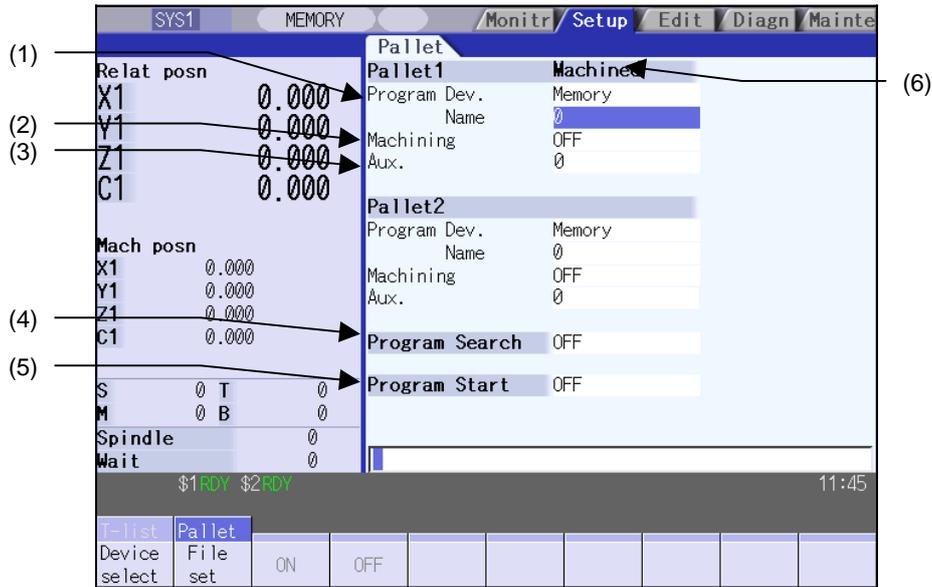
Screen		Function
Standard Pallet Registration screen		Registration of machining programs for pallet 1 & 2 is carried out. Pallet 1 & 2 can be registered at once.
Pallet 4 Page Registration screen	Pallet list screen	Machining program device, file name, machining status, and index planes under machining for 4 planes of the pallet 1 to 12 are displayed. Information of 2 pallets can be displayed at once.
	Pallet details screen	Machining program device, file name, machining status, and external workpiece coordinates for each plane of the pallet 1 to 12 are set. Information of one pallet's 2 indexing planes can be set at once.

(Note) The number of pallets is determined by the parameter "#11002 Valid pallet num".

#11001 APC type	Screen
0	Standard Pallet Registration
1	Pallet 4 Page Registration

Inputting an address key, such as MST, displays manual numerical value command window where manual numerical value command is executed.

3.13.1 Standard Pallet Registration



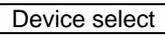
Display items

Display item	Details
(1) Program Dev./Name	The machining program device and file name registered for each pallet are displayed.
(2) Machining ON/OFF	The ON/OFF state of machining for each pallet is displayed.
(3) Aux.	If there are any auxiliary functions, the auxiliary function data for each pallet is displayed. Refer to the Instruction Manual issued by the machine tool builder for details on the corresponding functions.
(4) Program Search ON/OFF	The ON/OFF state of search for the registered machining program is displayed.
(5) Program Start ON/OFF	The ON/OFF state of continuous start for the registered machining program is displayed. When set to ON, the searched program will be started for the pallet in the machine. This setting is valid only when Pallet Program Search is set to ON.
(6) Machined	"Machined" is displayed for the pallet in the machine.

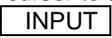
Menus

Menu	Details	Type	Reference
Device select	Device menu is displayed. If selected, the device name of the pallet currently indicated by the cursor is changed.	C	Registering a machining program for the pallet
File set	The file name to be registered can be selected from the list. A list of files of the currently selected device is displayed. This menu can be used only when the cursor is at file name.	C	Selecting a file name from the list
ON	The state (Machining, Program Search, Program Start) indicated by the cursor is validated. This menu can be used only when the cursor is at Machining, Program Search, or Program Start.	C	Registering a machining program for the pallet
OFF	The state (Machining, Program Search, Program Start) indicated by the cursor is invalidated. This menu can be used only when the cursor is at Machining, Program Search, or Program Start.	C	

Operation method (Registering a machining program for the pallet)

- (1) Using the , , ,  keys, move the cursor to the area of the pallet for which a program is to be registered.
- (2) Press the  menu and select a device.  The cursor is moved to "Device". The device menu (Memory, HD, DS, FD, Memory card) is displayed. If selected, the device selected in "Device" is displayed.
- (3) Move the cursor to "File name" and input the program No.  The contents set in "File name" is displayed.
- (4) Move the cursor to "Machining" and press the  or  menu.  The contents set in "Machining" is displayed.
- (5) Move the cursor to "Program Search" and press the  or  menu.  The contents set in "Program Search" is displayed.
- (6) Move the cursor to "Program Start" and press the ON or OFF menu.  The contents set in "Program Start" is displayed.

Operation method (Selecting a file from the list)

- (1) Move the cursor to "File name" and press the  menu.  A list of files for the specified device is displayed.
- (2) Move the cursor to an arbitrary file and press the  key.  The selected file name is displayed in "File name". The cursor is moved to "Machining". An arbitrary file name can be directly entered, as well.

(Note 1) When a file name is directly entered, whether or not the file exists is not checked.

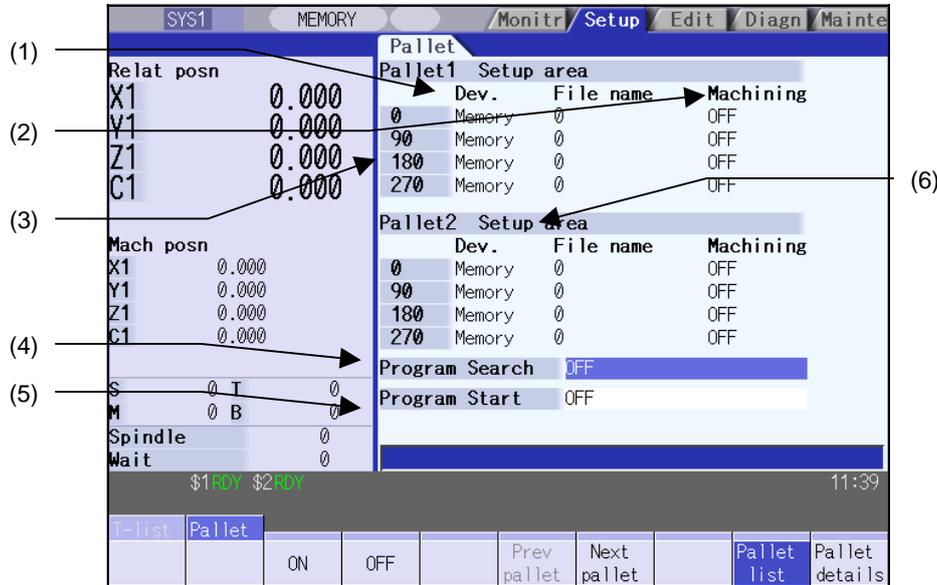
(Note 2) When a file name is directly entered, if other than character strings or values within the setting range (1 to 99,999,999) is set, an error occurs.

(Note 3) If a directory is specified, a setting error occurs.

3.13.2 Pallet List Screen

Machining program device, file name, machining status, and index planes under machining for 4 planes of the pallet 1 to 12 are displayed.

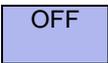
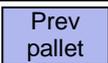
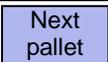
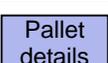
(Note) The number of pallets is determined by the parameter "#11002 Valid pallet num".



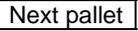
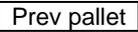
Display items

Display item	Details
(1) Dev./File name	Machining program device name and file name to be registered in each pallet's index plane are displayed.
(2) Machining ON/OFF	Whether machining for each pallet's index plane is valid or invalid is displayed.
(3) Machining index plane	0°, 90°, 180°, 270° are displayed. An asterisk (*) is displayed with the angle of the index plane.
(4) Program Search ON/OFF	Whether search for the registered machining program is valid or invalid is displayed. When valid, the next program registered will be searched automatically as soon as the currently machining program finishes. The next program is one of the valid index plane programs after the pallet in the machine. External workpiece coordinate set in the pallet detailed screen during search is set in the external workpiece coordinate shift (EXT). Auto restart is disabled. If auto restart is turned ON by PLC switch, it will be automatically turned OFF.
(5) Program Start ON/OFF	Whether continuous start of the registered program is valid or invalid is displayed. When valid, run the searched program for a pallet in the machine. This setting is valid only when pallet program search is valid.
(6) Pallet information in the machine	Whether pallet is in the machine is displayed. <ul style="list-style-type: none"> • Mach. area: When pallet is in the machine • Setup area: When pallet is not in the machine

Menus

Menu	Details	Type	Reference
	The state (Machining, Program Search, Program Start) indicated by the cursor is validated. This menu can be used only when the cursor is at Machining, Program Search, or Program Start.	C	Setting pallet program search
	The state (Machining, Program Search, Program Start) indicated by the cursor is invalidated. This menu can be used only when the cursor is at Machining, Program Search, or Program Start.	C	Setting continuous start
	Information on the previous two pallets is displayed.	C	Changing display pallet
	Information on the next two pallets is displayed.	C	
	A list of pallets is displayed.	B	
	The screen is shifted to the Pallet details screen. The 0° or 90° data of the pallet displayed on the top line of the Pallet list screen is displayed.	B	3.13.3 Pallet Details Screen

Operation method (Changing display pallet)

- (1) When the  are displayed, press the  menu or next page key.  Pallet 3 and 4 are displayed.
- (2) Press the  menu or previous page key.  Pallet 1 and 2 are displayed.

Operation method (Setting pallet program search)

- (1) Using the , , ,  keys, move the cursor to "Program Search" and press the  or  menu.  The contents set in "Program Search" is displayed. The cursor moves to "Program Start".

Operation method (Setting continuous start)

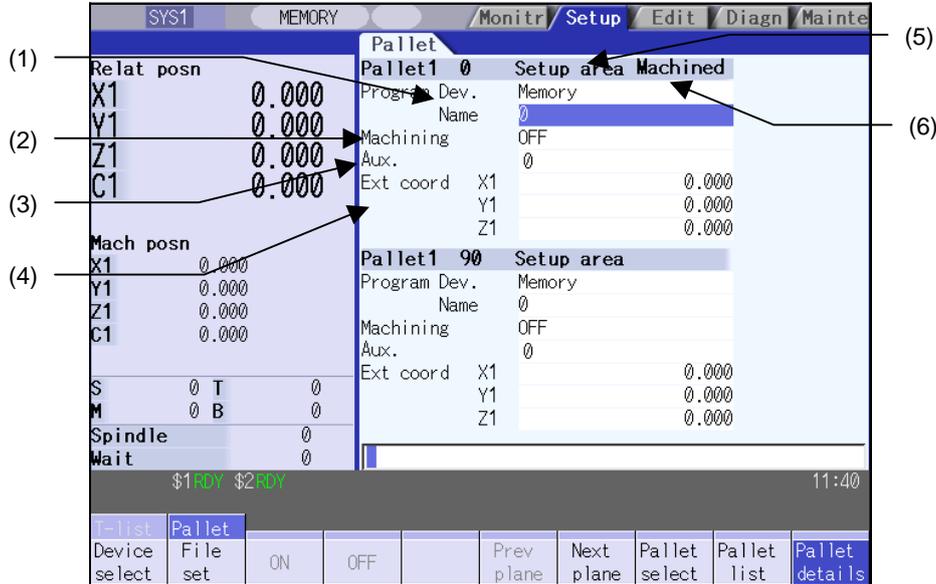
- (1) Using the , , ,  keys, move the cursor to "Program Start" and press the  or  menu.  The contents set in "Program Start" is displayed. The cursor moves to "Program Search".

Every time the , , , or  key is pressed, the cursor switches between "Program Search" and "Program Start".

3.13.3 Pallet Details Screen

Machining program device, file name, machining status, Aux. and external workpiece coordinates for each plane of the pallet 1 to 12 are set.

(Note) The number of pallets is determined by the parameter "#11002 Valid pallet num".



Display items

Display item	Details
(1) Program Dev./Name	Machining program device name and file name to be registered in each pallet's index plane are displayed.
(2) Machining ON/OFF	Whether machining for each pallet's index plane is valid or invalid is displayed.
(3) Aux.	If auxiliary function is available, data concerning auxiliary function for each pallet is displayed. Refer to the instruction manual issued by the machine tool builder for details on this function.
(4) Ext coord	<p>External workpiece coordinate for each pallet's index plane is set. The value set in the external workpiece coordinate is written into the external workpiece coordinate shift (EXT) according to the command from PLC. When the external workpiece coordinate for the machining surface of a pallet in machine is changed, only the changed axis data is written into the external workpiece coordinate shift (EXT). The set data (X to Z axis) is set in the pallet No. and extended workpiece coordinate offset corresponding to the index plane.</p> <p>X to Z axis: External workpiece coordinate data</p> <p>(Example)</p> <p>Pallet 1 0° : Extended workpiece coordinate offset 49 sets Pallet 1 90° : Extended workpiece coordinate offset 50 sets Pallet 1 180° : Extended workpiece coordinate offset 51 sets Pallet 1 270° : Extended workpiece coordinate offset 52 sets Pallet 2 0° : Extended workpiece coordinate offset 53 sets : Pallet 12 270° : Extended workpiece coordinate offset 96 sets</p>

3. Setup Screens

3.13 Pallet Program Registration [700 series only]

Display item	Details
(5) Pallet information in the machine	Whether pallet is in the machine is displayed. <ul style="list-style-type: none"> • Mach. area: When pallet is in the machine • Setup area: When pallet is not in the machine
(6) Machined	"Machined" is displayed on the index plane of a pallet in the machine.

(Note 1) The operation message "Pallet running" appears when the setting of "Program Dev.", "Program Name", "Machining", or "Aux." is executed to the index plane under machining of a pallet in machine. The "Program Dev.", "Program Name", "Machining", or "Aux." can be set when the index plane of a pallet in machine is not under machining. Even if the index plane of a pallet in machine is under machining, the external workpiece coordinate can be set.

Menus

Menu	Details	Type	Reference
Device select	The device menu is displayed. When selected, device name of the pallet at the cursor position is changed. When a device other than memory is selected, the route is displayed first.	C	Setting values in each index plane
File set	File name to be registered can be selected from a list. A list of files of the currently selected device is displayed. When the cursor is pointing at file name, this menu is operatable.	C	3.13.1 Standard Pallet Registration (Selecting a file from the list)
ON	The state (Machining) indicated by the cursor is validated. This menu can be used only when the cursor is at Machining.	C	Setting values in each index plane
OFF	The state (Machining) indicated by the cursor is invalidated. This menu can be used only when the cursor is at Machining.	C	
Prev plane	Previous index plane is displayed. When 0° or 90° index plane is being displayed, this menu displays the previous pallet, 180° or 270° index plane. When 180° or 270° index plane is being displayed, this menu displays 0° or 90° index plane. When 0° or 90° index plane of the pallet 1 is displayed, this menu cannot be selected.	C	Changing index planes
Next plane	Next index plane is displayed. When 0° or 90° index plane is being displayed, this menu displays 180° or 270° index plane. When 180° or 270° index plane is being displayed, this menu displays the next pallet, 0° or 90° index plane. When 180° or 270° index plane of the last pallet is displayed, this menu cannot be selected.	C	
Pallet select	An arbitrary pallet can be selected. When the pallet No. is set and the [INPUT] key is pressed, the 0° or 90° pallet details data for that pallet will be displayed. The cursor will move to the 0° machining program file name.	A	Changing display pallets
Pallet list	The Pallet list screen will open. A list of pallets, including the pallet displayed on the Pallet details screen will appear.	B	3.13.1 Standard Pallet Registration
Pallet details	This displays the pallet details.	B	

Operation method (Changing display pallets)

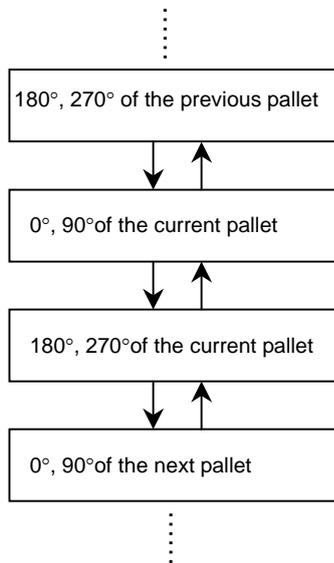
- (1) Press the **Pallet select** menu. Input pallet No. and press the **INPUT** key.  The pallet 5's 0° and 90° index plane will appear.

Operation method (Changing index planes)

(Example) When the screen display status acquired after the above (Changing display pallets) operation

- (1) Press the **Next plane** menu or next page key.  The pallet 5's 180° and 270° index plane will appear.
- (2) Press the **Prev plane** or previous page key.  The pallet 5's 0° and 90° index plane will appear.

The display of pallet and index plane changes as follows.



Operation method (Setting values in each index plane)

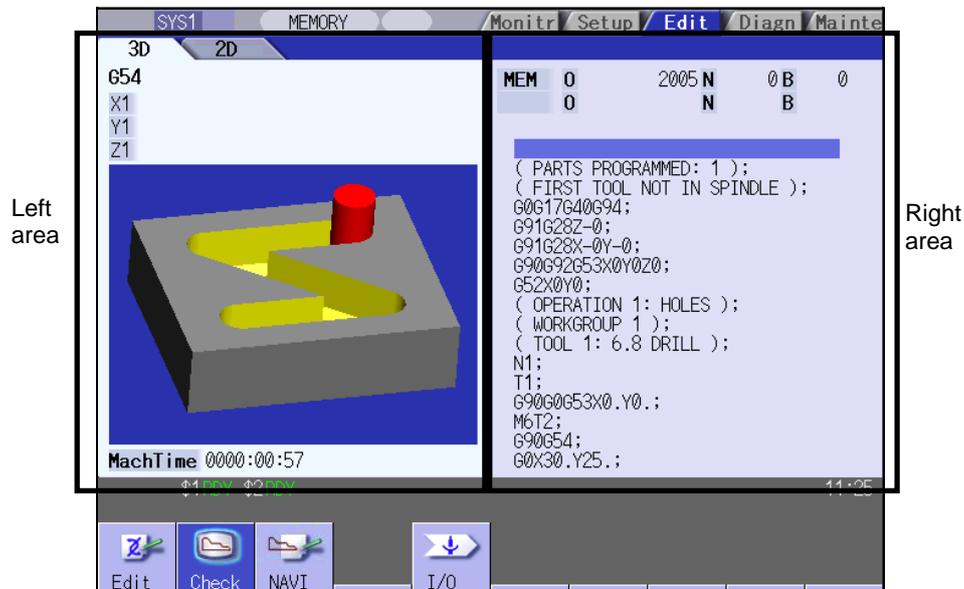
- (1) An arbitrary index plane is displayed using the operation method above.
- (2) Press the menu and select a device.  The cursor moves to "Device". The device menu (Memory, HD, DS, FD, Memory card) is displayed. If selected, the device selected in "Device" is displayed. After setting, the cursor moves to "File name".
- (3) Enter program No. in "File name".  Contents set in "File name" is displayed. The cursor moves to "Machining". A file can be selected from the list, as well.
- (4) Press the or menu for "Machining".  The contents set in "Machining" is displayed. The cursor moves down.
- (5) Move the cursor to "Ext coord" and input values for 3 axes.  Values for 3 axes are set. The cursor moves to the "Program File name" above.

4. Edit Screens

The Edit screen is used to edit the machining program (adding, erasing and changing), perform checks, and input/output data.

4.1 Screen Configuration

The Edit screen is configured as shown below.



Left area :The contents differ depending on the display type.

Use the **Display change** menu or **←**, **→** key to switch the display type.

The tag at the upper section of the area depends on the option validity.

Right area: Displays the program searched to be edited.

When the multi-program display type is applied, use **<--> change** menu key to switch the active area.

Menus

Menu	Details	Type	Reference
 Edit	This edits the machining program.		4.2 Program Editing
 Check	This checks the program. This is used to check the machining program without performing automatic operation. (1) Program check (2D) This displays a graphic drawing of the machining program movement path. (2) 3D solid graphic check This displays a solid graphic drawing of the workpiece shape and tool movement during the cutting process in the machining program. This menu does not display if there is no program check function option.		4.3 Program Check (2D) 4.4 Program Check (3D)
 NAVI	This creates the part program simply. Refer to the following manuals for detail. 700/70 Series Simple Programming Function NAVI MILL Instruction Manual (IB-1500144(ENG)) 700/70 Series Simple Programming Function NAVI LATHE Instruction Manual (IB-1500146(ENG))		
 I/O	This performs the machining program input and output between the NC internal memory and the external input/output device. (Note) When "#8923 Hide Edit-IO menu" is "1", this menu is not displayed.		4.5 Program Input/Output

4.2 Program Editing



This is used to edit (add, erase, change) the machining programs in the NC memory, HD, memory card (front IC card), DS (compact flash) or FD, and to create new programs. (In 70 series, only the memory and the memory card can be used.)

This function is used for three types of program: machining program, MDI program and fixed cycle program. Press the main menu **Edit** key to display the program found when performing an operation search (MDI program in MDI mode).

For only the machining programs in NC memory, the program for each part system can be saved/edited by one program name when the multi-part system program management is valid (#1285 ext21/bit0 =1).

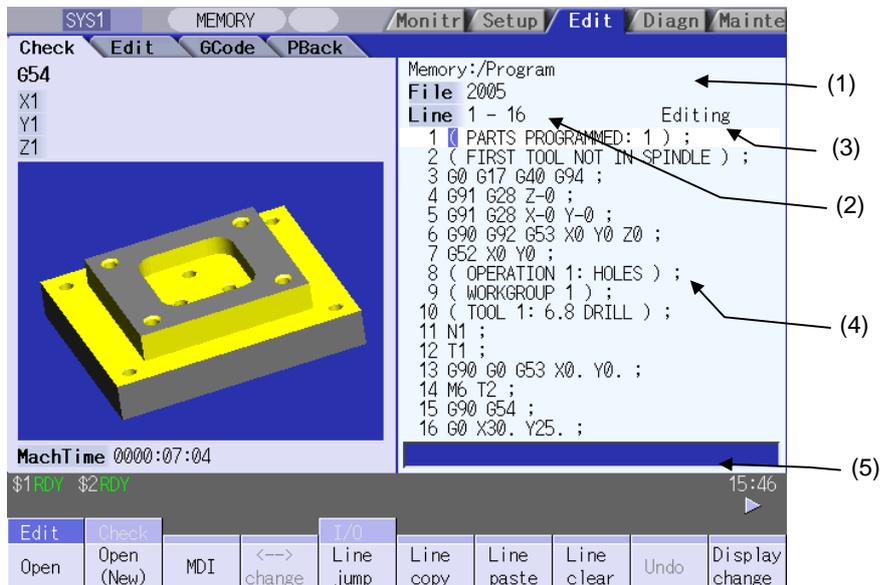
The contents of the left area depends on the display type:

- Check display type :Enables to edit a program confirming the program check screen.
- Multi-program display type :Enables to edit two programs using left and right editing areas.
- G-code guidance display type :Enables to edit a program referring to the G code guidance.
- Playback display type :Enables to edit program trying sample machining.

There are two types of editing available: regular editing and mass-editing. The specification and the limitation are different between regular editing and mass-editing.

The conditions and the maximum edit size of mass-editing are as follows.

NC model	Conditions of mass-editing	Maximum edit size
700 series	<ul style="list-style-type: none"> • Storage location of the opened machining program is either HD, FD, memory card or DS. • The size of the file is 1.0MB or more. (Except when parameter "#8910 Edit Undo" is "0", which needs 2.0MB or more.) 	FCU7-DA2-xx : 20MB FCU7-DA3-xx/DA4-xx : 1GB
70 series	<ul style="list-style-type: none"> • Storage location of the opened machining program is memory card. • The size of the file is 0.5MB or more. 	10MB



Display items

Display item	Details
(1) Path display	This displays the path for the open program file. (Ex.) Memory: /program When the path is too long, the characters exceeding the 37 characters (1-byte code) can not be displayed.
(2) Program name Top display line Last display line	[Program name (file)] This displays the file name of the program currently being edited. This displays "MDI" when the MDI program name is being edited. [Top display line, Last display line] This displays the line number for the first and last line of the currently displayed program. Proving an entire program is regarded as 100%, the position of the page currently displayed is shown in percentage during mass-editing.
(3) N No. additional value display Edit type display Current edit display "Editing" Insert mode display "INS"	[N No. additional value display] This displays an automatic additional value of sequence No. [Edit type display] This displays "EX" in mass-editing. [Current edit display "Editing"] This displays when performing edit operations after displaying the program. [Insert mode display "INS"] This displays when pressing the <input type="button" value="INSERT"/> key and changing to the insert mode.
(4) Line number Program display	[Line number] This displays the last 3 digits of the program line number. The line No. attaches only to the first line when over two or more lines are displayed on the screen because one line is long. The line No. does not attach during mass-editing. [Program display] This displays the contents of the program (machining program, MDI program) currently being edited. The line with the cursor is emphasized. The line is not emphasized during mass-editing.
(5) Input area	This inputs the program line number and search character string.

Menus

Menu	Details	Type	Reference
<input type="button" value="Open"/>	This edits and references the existing programs. When the program number is designated and the <input type="button" value="INPUT"/> key is pressed, the program contents appear and can be edited.	A	4.2.2 Editing a Machining Program
<input type="button" value="Open (New)"/>	This creates a new program. When the program number is set and the <input type="button" value="INPUT"/> key is pressed, a new program is created.	A	4.2.1 Creating a New Machining Program
<input type="button" value="MDI"/>	This edits the MDI program. When this menu key is pressed, the MDI program appears and can be edited.	A	4.2.3 Editing MDI Program
<input type="button" value="↔ change"/>	This switches the active area between left and right editing areas. Editing can be done in the active area. This menu can be used with the multi-program display type.	C	4.2.7 Changing the Display
<input type="button" value="Line jump"/>	With regular editing, when the line number is set and the <input type="button" value="INPUT"/> key is pressed, the cursor moves to that line number.	A	4.2.8 Designating an Arbitrary Line
<input type="button" value="% jump"/>	With mass-editing, proving an entire program is regarded as 100%, designate the position of the page to be displayed in percentage and press the <input type="button" value="INPUT"/> key. The program data of the designated display position is displayed on the screen. The cursor jumps to the top line of the displayed program.		

Menu	Details	Type	Reference
Line copy	This copies the designated line (multiple lines can be copied.) This menu cannot be selected during mass-editing.	A	4.2.14 Copying/Pasting Data
Line paste	This inserts the copied line at the line before the cursor. This menu cannot be selected during mass-editing.	C	
Line clear	This erases the designated line (multiple lines can be erased.) This menu cannot be selected during mass-editing.	A	4.2.11 Deleting Data
Undo	This returns the contents of the program to that prior to saving and display. This menu cannot be selected when parameter "#8910 Edit Undo" is set to "0". This menu cannot be selected during mass-editing. In 70 series, this menu cannot be selected.	C	4.2.15 Undoing Program Changes
Display change	This switches the display type.	C	
Comment nondisp	This changes whether to show or hide the comment field in the list. When the comment field is hidden, the file name field will be enlarged.	B	
String search	When the character string is designated and the <input type="text" value="INPUT"/> key is pressed, that character string is searched for. This menu cannot be selected during mass-editing.	A	4.2.12 Searching for Character Strings
String replace	If the character string to be searched for and the character string to be replaced are separated with a "/" and designated, when the <input type="text" value="INPUT"/> key is pressed, the character string is replaced. This menu cannot be selected during mass-editing.	A	4.2.13 Replacing Character Strings
Miss warning	This enables the input mistake check for the program being edited. This menu cannot be selected during mass-editing. This menu is only for 700 series.	B	4.2.16 Correcting/Displaying Input Mistakes
Next miss	This moves the cursor to the next input mistake warning location. This menu cannot be selected during mass-editing. This menu is only for 700 series.	C	
N auto add	The sequence No. can be added automatically by pressing the <input type="text" value="INPUT"/> key after designating an additional value. This menu cannot be selected during mass-editing.	A	4.2.17 Adding Sequence No. (N No.) Automatically
MDI regist	This registers the MDI program in the memory. This can be performed only when the MDI program is displayed.	A	4.2.3 Editing MDI Program
Erase file	This deletes the programs. When the name of the program to be erased is designated and the <input type="text" value="INPUT"/> key is pressed, the designated file is deleted.	A	4.2.5 Deleting a File

4.2.1 Creating a New Machining Program

Operation method

- (1) Press the menu . → The following menu appears.
The list appears as a pop-up window.

Memory	HD		Memory card	DS	FD		List update	Sort change	Close
--------	----	--	-------------	----	----	--	-------------	-------------	-------

(When specifications of all devices is valid.)

- (2) Select a device.
(Ex.) Menu → The selected device name and directory (Memory: /Program) appear in the Device name, Directory display column.
For the devices other than memory, root directory is selected.

- (3) Input the file name for the program to be newly created and press the key.
(Ex.) 100 → If a program can be newly created, a program containing only EOR is created.
The list display is closed.

```
Memory:/Program
File 100
Line 1 - 1
1
```

- (4) Edit a machining program. → Refer to "4.2.6 Edit Operations" and later.

- (5) Press the key. → The created machining program is saved to the device.

(Note 1) An error occurs if an existing program No. is set.

(Note 2) The texts in parentheses () at the top block of the program is the comment.

(Note 3) The existing file cannot be set.

(Note 4) The characters that can be used for the file name and directory path are 1-byte number, 1-byte capital alphabetical letters, and 1-byte symbols recognized by the system.

Note that the following characters cannot be used.

|, /, :, ", " (comma), *, ?, ", <, >, a to z (small letters), Space

In addition, the following cases cannot be treated as the file name.

- When the extension is such as "\$\$\$", "\$\$0", "\$\$1", "\$\$2", "\$\$3", "\$\$4", "\$\$5", "\$\$6", "\$\$7", "\$\$8", "\$\$9".
- When the file name is 1-byte character "0" (zero).

(Note 5) Creation of a program is not possible for a program with 33 or more file name characters.

4.2.2 Editing a Machining Program

Operation method

- (1) Press the menu. → The following menu appears.
The list appears as a pop-up window.

Memory	HD		Memory card	DS	FD		List update	Sort change	Close
--------	----	--	----------------	----	----	--	----------------	----------------	-------

(When specifications of all devices is valid.)

- (2) Select a device.
(Ex.) Menu → The selected device name and directory (memory:
/program) appear in the Device name, Directory
display column.
For the devices other than memory, root directory is
selected.

- (3) Use the , , , keys to align
the cursor with target machining program.
The name of the machining program to be
edited can be input in the input area.

- (4) Press the key. → The program appears at the top if the file can be
opened.
The cursor moves to the head character of the
program. The mode enters the overwrite mode.
The list display is closed.

```
Memory:/Program
File 348
Line 1 - 16
1 G348(PARTS1) ;
2 G28 XYZ;
3 G92 G53 XYZ;
4 T3 ;
5 M5 T5 ;
6 N10 ;
7 G90 G0 X0 Y0 ;
8 Z5 ;
9 G343 H3 Z5. ;
10 M8 ;
11 F1000 S3000 ;
12 M3 ;
13 N20 ;
14 G41 D3 ;
15 G3 Z-20. I20. P3. ;
16 G1 X50. Y100. F1000 ;
```

- (5) Edit a machining program. → Refer to "4.2.6 Edit Operations" and later.
- (6) Press the key. → The edited machining program is saved to the device.

(Note 1) An error occurs if a nonexistent program No. is set.

(Note 2) If the selected program is running or program restarting, it can be displayed but not edited.
An error occurs when the data is set or the key is pressed.

(Note 3) The fixed cycle program list displays when basic common parameter "#1166 fixpro" is set to "1".

(Note 4) Editable size differs depending on each device. When the editable size is exceeded, the
message "Can't edit because of size over" appears.

(Note 5) "Loading" is displayed in flickering until the file is opened.

(Note 6) Editing is not possible for a program with 33 or more file name characters.

(Note 7) When the program in search was edited, the program is searched again as follows depending
on the last operation.

Search -> Edit : The position in which ONB is displayed is searched again.

Search -> Reset 1-> Edit : The program top is searched again.

Search -> Reset 2-> Edit : The search is not executed. If a program was added or
deleted, the search position may shift.

Search -> Restart & Search -> Edit: The program top is searched again.

4.2.3 Editing MDI Program

Operation method

- (1) Press the menu . → The menu is highlighted.
The MDI program is displayed from the head of the active editing area. The cursor moves to the head character of the program.
The mode enters the overwrite mode.

```
Memory:/Program
File MDI
Line 1 - 4
1 G28 XYZ;
2 G32 G53 XYZ;
3 G40 G49 G80 G89 ;
4 %
```

- (2) Edit the MDI program. → Refer to "4.2.6 Edit Operations" and later.

(Note 1) Before starting MDI operation, confirm that the MDI settings are completed. If "Editing" or "MDI no setting" is displayed, the MDI operation cannot be started.

When the key is pressed, the head block will be searched, and the message "MDI search complete" will appear. The setting will then be completed.

If the cursor is moved without editing, the block indicated by the cursor will be searched for when the key is pressed.

(Note 2) If the key is pressed when the number of MDI program characters including EOB (;) and EOR (%) exceeds 2000, "Memory capacity over" message will appear and the MDI program will not be searched for execution.

Note that the contents edited after the MDI program is saved last will not be saved in the NC memory.

4.2.4 Registering MDI Program in NC Memory

Operation method

- (1) Press the menu . → The menu is highlighted and turned to the program No. input wait status.
The cursor appears in the input area.

- (2) Set the program No. to be registered and press the key. → MDI program is registered in the memory. The operation message "MDI entry complete" is displayed.
The menu highlight returns to normal. The cursor returns to the program display area.

(Note 1) When the key is pressed without entering the program name, a setting error occurs.

(Note 2) When the entered program name already exists in the NC memory, the operation message "Overwrite this file?(Y/N)" is displayed.

(Note 3) MDI programs cannot be registered in the NC memory in the following cases.

- When the setting is made for other than programs with the parameter "#1166 fixpro".
- During "Editing" is displayed.
- When the remaining size of program is smaller than the MDI program to be registered.
- When the remaining number of programs that can be registered is "0".
- Entered program name already exists in the NC memory, and the status is in automatic operation or program restarting.

4.2.5 Deleting a File

Operation method

- (1) Press the menu → The following menu appears.
The list appears as a pop-up window.

Memory	HD		Memory card	DS	FD		List update	Sort change	Close
--------	----	--	----------------	----	----	--	----------------	----------------	-------

(When specifications of all devices is valid.)

- (2) Select a device.
(Ex.) Menu → The selected device name and directory (memory: /program) appear in the Device name, Directory display column.
For the devices other than memory, root directory is selected.

- (3) Use the , , , keys to align the cursor with target machining program.
The name of the machining program to be deleted can be input in the input area.

- (4) Press the key. → The operation message "Erase? (Y/N)" appears.

- (5) Press the or key. → The file is deleted.
Press the key not to delete. → The list display is closed.

(Note) The file is not deleted in the following cases.

- The file that is to be deleted is currently being used in automatic operation.
- The file that is to be deleted is subject to edit lock B or C.
- Data protection key 3 is enabled.
- The file to be deleted is in the "program restarting" state.

4.2.6 Editing Operations

When the program is edited, the key input data is directly written into the program display area. All data is overwritten from the cursor position. "Editing" appears on the right side of the file name display once input is started.

Press the key to save the program to the device and clear the "editing" message.

The editing operations from when the file is actually opened are explained in the following sections.

These editing operations are common for the machining program and MDI program.

CAUTION

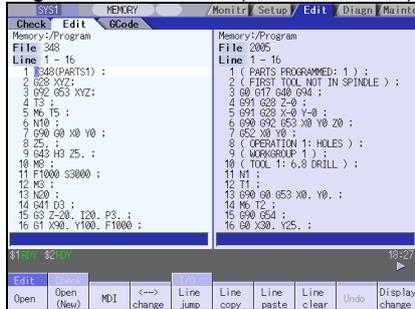
-  If "there is no value after the G" command, the operation will be the "G00" operation when the program is run due to key chattering, etc., during editing.

4.2.7 Changing the Display

Switching the active area

Switching the active area is possible only when the multi-program display type is selected. Press the **<--> change** key to switch the active area between the left and right window.

Right side is active (can be edited)



Left side is active (can be edited)



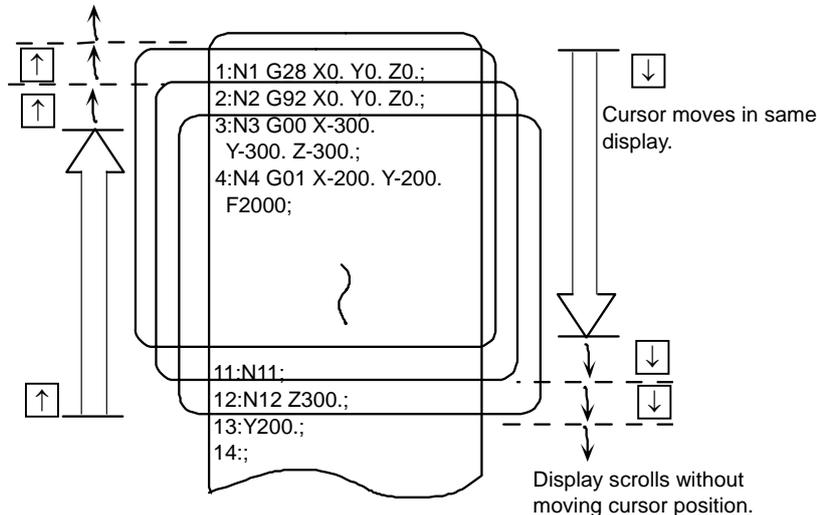
<--> change

Changing the display using the page changeover keys (**▲**: Previous Page, **▼**: Next Page)

- ▲** key: This displays one page of preceding lines of the current top line. When there is less than one page of data, a page of data will be displayed including the data currently displayed. (The cursor moves to the last line of the screen. Cursor does not move in mass-editing.)
- ▼** key: This displays one page of following lines of the current bottom line. (The cursor moves to the top line of the screen. Cursor does not move in mass-editing.)

Changing the display with the cursor key

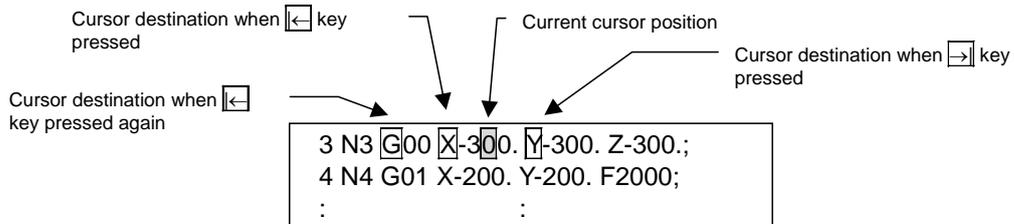
The cursor will move up one line in the program each time the cursor key (**↑**, **↓**) is pressed. If the **↓** key is pressed at the top line of the program display area, the program will scroll up one line. If the **↑** key is pressed at the end, the program will scroll down one line. When a block is displayed over 2 or more lines, **↑**, **↓** keys move the cursor by block unit during regular editing and by line unit during mass-editing.



Move the cursor using the tab keys (: back tab/ : tab).

key : Moves to the start of the word where the cursor is currently positioned.
If the cursor is at the start of a word, it moves to the start of the previous word.

key : Moves to the word after the word where the cursor is currently positioned.



(Note) Pressing keys or keys does not make any difference in mass-editing.

Split display by word unit

In regular editing, program is split by word unit and displayed.
In mass-editing, program is not split by word unit and displayed as text data.

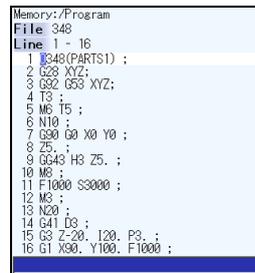
4.2.8 Displaying an Arbitrary Line

Operation method

(1) Press the menu .



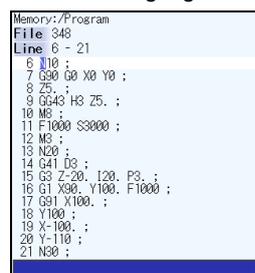
The menu is highlighted.
The cursor appears at the lower input area.



(2) Input the line No.
(Ex.) 6



The program appears with the set line No. at the head.
The cursor moves to the head.
The menu highlight returns to normal.



(Note 1) When "0" is input, the cursor moves to the first line.

(Note 2) When "E" is input, the cursor moves to the end.

4.2.9 Rewriting Data

Operation method

- (1) Move the cursor to the position of the data to be rewritten.
The cursor can be moved to one place to the right of EOB (;).
- (2) Set the data.  The message "Editing" appears when setting is started.
The data is set from the position of the cursor. Data at the area is overwritten.
The cursor moves one space (character) at a time to the right as the data is set.
- (3) When one line of setting is completed, press the INPUT key.  EOB (;) is added to the set data and the data is fixed.
When newly creating data, the cursor moves to the head of the next line.
When changing existing data, etc., the cursor does not move.

- (Note 1)** Up to 255 characters can be input in one block in regular editing and 127 characters in mass-editing.
- (Note 2)** When the cursor is on or one place to the right of EOB (;), input data is inserted even if not in input mode (refer to "4.2.10 Inserting Data".)
- (Note 3)** When changing to another function such as the program check, or to another screen such as the operation screen when "Editing" displays, the operation message "Save current file? (Y/N)" displays. If N key is pressed changing to the other screen once, and the Edit screen is selected again, the program before editing operation will be displayed. (Edited contents will be invalid.)
- (Note 4)** "Saving" is displayed in flickering during saving.

4.2.10 Inserting Data

Operation method

- (1) Move the cursor to the position where data is to be inserted.
- (2) Press the INSERT key.  The insertion mode is entered. "INS" and "Editing" appear to the right of the file name.
- (3) Set the data.  The data is inserted before the cursor position. Data after the cursor moves to the right.
- (4) When the setting is completed, press the INPUT key.  The set data is fixed. The cursor does not move, but if there is no EOB (;) at the end of the line, it will be added. Then, the cursor moves to the next line. The mode returns to the overwrite mode, and the message "Editing" disappears.

(Note 1) Up to 256 characters can be set in one line.

(Note 2) The insertion mode will be finished if a key such as DELETE, C.B, CAN, INPUT, ↑, ↓, ←, →, |←, →|, ↑ or ↓ pressed.

(Note 3) During regular editing, EOB (;) is added to the line without EOB (;) before moving to the next line; EOB (;) is not added to the the line with EOB (;). (No cursor movement.)

(Note 4) During mass-editing, linefeed is inserted every time EOB (;) is input. (Cursor moves to the head of next line.)

4.2.11 Deleting Data

Operation method (Deleting one character)

- (1) Move the cursor to the character to be deleted.
- (2) Press the key.  1 character is deleted at the cursor position, and "Editing" appears. The data from the cursor position moves to the left. When this key is held down, the characters can be deleted one at a time.
- (3) Press the key.  The edited machining program is saved to the device and "Editing" disappears.

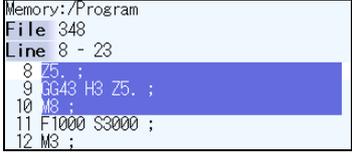
Operation method (Deleting one line)

- (1) Move the cursor to the line to be deleted.
- (2) Press the key.  1 line is deleted at the cursor position, and "Editing" appears. The lines following the cursor position are shifted upward. The cursor position does not change. When this key is held down, the line can be deleted one at a time.
- (3) Press the key.  The edited machining program is saved to the device and "Editing" disappears.

(Note 1) In the case where the block for the deleted line covers multiple lines, these lines are also deleted.

(Note 2) Last line only with "%" cannot be deleted.

Operation method (Deleting designated lines)

- (1) Press the menu. → The menu is highlighted.
The cursor appears at the lower input area.
- (2) Designate the deletion range.
(Ex.) From line 8 to line 10
8/10 → The background color for the range to be deleted
turns light blue.
The message "Erase? (Y/N) " appears.
- When deleting 1 line, it is also possible to
make a selection using the , keys
and designate by pressing the
key.
- 
- ```
Memory:/Program
File 348
Line 8 - 23
8 Z5. ;
9 G343 H3 Z5. ;
10 M3 ;
11 F1000 S3000 ;
12 M3 ;
```
- (3) Press the  or  key. → The data in the range with the blue background is  
deleted and the menu highlight returns to normal.  
The lines following the deleted data are shifted  
upward.  
The cursor position and top line No. do not change.  
Press the  key not to delete.

**(Note 1)** If the  key is pressed without setting a line No., the line where the cursor is will be deleted.

**(Note 2)** If the last line of the file is the range to be deleted, the last line can be designated as "E".

**(Example)** From 8th line to last line : 8/E  
From 1st line to last line: /E

**(Note 3)** Mass-editing has restrictions below.

- The range has to be within 100 lines.
- Last line designation switch "/E" is invalid.

**(Note 4)** "Executing" is displayed in flickering during line clear.

## Operation method (Deleting several lines (one screen of data) displayed on a screen)

- (1) Press the  and  keys. → Data (machining program) for 1 screen is deleted, and  
"Editing" appears.
- (2) Press the  key. → The edited machining program is saved to the device  
and "Editing" disappears.

## 4.2.12 Searching for Character Strings

## Operation method

- (1) Press the  menu. → The menu is highlighted.  
The cursor appears at the lower input area.
- (2) Set the character string to be searched for, and press the  key.  
(Ex.) G20  → A search for a character string is made from the character after the cursor position downwards.  
If the character string is discovered, the cursor moves to the start of that character string.  
A message appears if there is no character string applicable to the program.  
Marking can be performed by adding an option to the character string being searched for.  
(Ex.) G20/MR :G20 are marked in red.  
(Refer to the "Marking function" for further details.)
- (3) To continue searching, press the  key again. → The next corresponding character string is searched for.  
The search finishes when the end of the program is reached.  
**(Note)** It is not possible to return to the head of the program and resume the search. To search from the head of the program, move the cursor to the top line and carry out the search operation again.
- (4) To end the search, press the menu key. → The menu highlight returns to normal, and the character string in the input area disappears.  
The search mode is held until the menu key is pressed.  
(The character string remains in the input area.)

**(Note 1)** "Executing" is displayed in flickering during string search.

### Marking function

Marking can be performed by adding an option to the character string being searched for.

| Argument character string format | Details                                                                                                                | Marking color |
|----------------------------------|------------------------------------------------------------------------------------------------------------------------|---------------|
| "Search character string"        | The cursor moves to the character string searched for.                                                                 | None          |
| "Search character string/MR"     | The cursor moves to the character string searched for.<br>All displayed searched character strings are marked in red.  | Red           |
| "Search character string/MB"     | The cursor moves to the character string searched for.<br>All displayed searched character strings are marked in blue. | Blue          |
| "Search character string/MG"     | The cursor moves to the character string searched for.<br>All displayed searched character string are marked in green. | Green         |
| "/MC"                            | The marking for all marked character strings is cleared.                                                               | -             |
| "/MCR"                           | The marking for character strings marked in red is cleared.                                                            | -             |
| "/MCB"                           | The marking for character strings marked in blue is cleared.                                                           | -             |
| "/MCG"                           | The marking for character strings marked in green is cleared.                                                          | -             |

**(Note 1)** The marking status is maintained when another program is opened while character strings are still marked.

**(Note 2)** If the input mistake check warning function is enabled while character strings are still marked, the marking is given priority in the case where the input mistake location and the marked display overlap.

**(Note 3)** In the case where the search marking and input mistake check warning function are displayed simultaneously, the marking is cleared and the input mistake warning display remains.

**(Note 4)** With the multi-program display type, the search and marking is valid only in the area where the search and marking is carried out.

**(Note 5)** The marked status is maintained until the NC power is turned OFF.

**(Note 6)** Marking function is invalid in mass-editing.

### Important points when performing search

- (1) If the designated character string cannot be found, a "Designated character string not found" message displays.
- (2) A search is performed for a character string containing the designated character string, irrespective of the characters before and after. As a result, if "G2" is designated, G20 - G29 and G200 etc. will also be the subject of a search.

### Character string data setting example

| Designated character string | Search character string example                                        |
|-----------------------------|------------------------------------------------------------------------|
| N10                         | Character strings that include "N10" such as "N10" and "N100".         |
| N10 X100.                   | Character string "N10 X100".                                           |
| x-012.34                    | Character string "X-012.34" (A search is not performed for "X-12.34".) |

## 4.2.13 Replacing Character Strings

A character string in a program can be searched for and replaced.  
The following two methods can be used.

- The applicable character string is searched for and replaced one by one. (To search for the next character string, press the  key.)
- All applicable character strings in the program are replaced in a batch. (Designate "/G" when setting.)

## Operation method

- |                                                                                                                                                                                                                           |   |                                                                                                                                                                                                                                                                                                                                                         |
|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|---|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| (1) <input type="text" value="String replace"/> .                                                                                                                                                                         | ➔ | The menu is highlighted.<br>The cursor appears at the lower input area.                                                                                                                                                                                                                                                                                 |
| (2) Designate the search character string and the replace character string.<br>((Ex.) Search character string: G02<br>Replace character string: In the case of G03<br>G02/G03 <input type="text" value="INPUT"/> (Note 1) | ➔ | The character string is searched downward from the character after the current cursor position.<br>If a character string is discovered, the cursor moves to the start of that character string, and a "Replace? (Y/N)" message appears.<br>If any character string is not found in the program, the cursor does not move.                               |
| (3) Press the <input type="text" value="Y"/> or <input type="text" value="INPUT"/> key.<br>Press the <input type="text" value="N"/> key not to replace.                                                                   | ➔ | The character string is replaced, and a search is performed for the next applicable character string. (If a batch replacement is performed, a search is not made as the replace has been completed.)<br>A "Designated character string not found" message appears when the end of the program is reached.<br>"Editing" appears if replace is performed. |
| (4) Press the <input type="text" value="Y"/> or <input type="text" value="INPUT"/> key again if continuing the search/replace.                                                                                            | ➔ | The same operation as above is performed.                                                                                                                                                                                                                                                                                                               |
| (5) To end search/replace, press the <input type="text" value="String replace"/> .                                                                                                                                        | ➔ | The menu highlight returns to normal, and the character string in the input area disappears.<br>The replacement mode is held until the menu key is pressed.<br>(The character string remains in the setting area.)                                                                                                                                      |

- (Note 1)** When batch replacing for all the character strings in the program, add "/G" to the above setting.  
Search character string/replace character string/G (Ex.) G02/G03/G  
During batch replacement, after replacing all character strings to the end of the program, the process ends while showing the last character string replaced. Note that the replacement mode is held, and the character string remains in the input area.
- (Note 2)** If the program character strings from the current cursor position to the last line are to be replaced in a batch, add "/E" to the above setting.  
Search character string/replace character string/E (Ex.) G02/G03/E
- (Note 3)** "Executing" is displayed in flickering during string replace.

## 4.2.14 Copying/Pasting Data

## Operation method (Copying)

- (1) Press the  menu. → The menu is highlighted.  
The cursor appears at the lower input area.
- (2) Designate the range to be copied.  
(Ex.) From line 8 to line 10  
8/10  → The background color for the range to be copied turns light blue.  
The menu highlight returns to normal.
- When copying 1 line, it is also possible to make a selection using the ,  keys and designate by pressing the  key.

```
Memory:/Program
File 348
Line 8 - 23
8 Z5. ;
9 G643 H3 Z5. ;
10 M3 ;
11 F1000 S3000 ;
12 M3 ;
```

## Notes when copying

- (1) If  key is input without setting a line No., the line where the cursor is at is copied.
- (2) The highlight of the copy target lines disappears when editing operations are resumed.
- (3) If the file is edited after the Line copy menu is pressed, the data copied in the line is canceled.
- (4) If the last line of the file is the range to be copied, the last line can be designated as "E".  
(Example) From 8th line to last line : 8/E
- (5) The copied data is maintained while the copy source file is opened even if the display type or editing area is switched to another one.
- (6) Up to 100 lines can be copied in mass-editing.

## Operation method (Pasting the copied data)

- (1) Move the cursor to the line where the data is to be pasted.
- (2) Press the  menu. → The copied data is inserted in the line before the cursor position, and "Editing" appears.
- (3) Press the  key. → The edited machining program is saved to device, and "Editing" disappears.

**(Note 1)** With multi-program display type, data can be copied and pasted between both left and right areas.

**(Note 2)** Copy/past is not possible in between regular editing and mass-editing.

**(Note 3)** "Executing" is displayed in flickering during line paste.

## 4.2.15 Undoing Program Changes

This is used to return the program to the form when  was last pressed to save. This operation is valid for the operation of "Rewriting program", "Insert", "Delete", "Line Paste", "Line Clear", "Character String Replace" and "Undo" functions.

## Operation method (Pasting the copied data)

- (1) Move the cursor to the position to be corrected, and input a character.  
(Ex.) 135
- ➡ The data is replaced with input data and "Editing" appears.
- ```
Memory:/Program
File 348                               Editing
Line 6 - 21
6 N10 ;
7 G90 G0 X0 Y0 ;
8 Z135.
9 G643 H3 Z5. ;
10 M3 ;
```
- (2) Press the key.
- ➡ The edited machining program is saved to the device and "Editing" disappears.
- (3) Press the menu .
- ➡ The contents of the file return to the same condition as when the editing was started. These contents of the file are saved to the device.
- (4) Press the menu again.
- ➡ The contents of the file return to the same condition as when the file was saved in the procedure (2). These contents of the file are saved to the device.

(Note1) This function is only valid when parameter "#8910 Edit Undo" is set to "1".

(Note2) When returned to the previous status, the cursor will move to the head line.

(Note3) With the multi-program display type, returning to the previous contents is valid only in the active area.

(Note 4) Undoing is not possible in mass-editing mode.

(Note 5) "Executing" is displayed in flickering during undoing.

4.2.16 Correcting/Displaying Input Mistakes [700 series only]

The input mistake check warning function regards the following cases as input mistakes and displays a warning.

Check Item	Warning display range	Details
No decimal point input	Applicable address and data	The decimal point was omitted from the data. (Ex.) N01 G0 X100 Y50.; "X100" displayed as warning

The addresses to check no decimal point input are shown below.

(○: Check performed, -: No check performed)

Address	Decimal point command	Application	M/L	Decimal point check
A	Enabled	Coordinate position data	M L L L	○ Only for axis name
	Disabled	Miscellaneous function code		
	Enabled	Geometric linear angle data		
	Disabled	Rotation table		
	Disabled	MRC program number		
	Disabled	Data settings, axis number		
	Enabled	Deep hole drilling cycle (2) Safety distance		
B	Enabled	Coordinate position data	M	○ Only for axis name
	Disabled	Miscellaneous function code		
	Disabled	Rotation table		
C	Enabled	Coordinate position data	M L	○ Only for axis name
	Disabled	Miscellaneous function code		
	Enabled	Corner chamfering amount (.C)		
	Disabled	Rotation table		
	Enabled	Program tool compensation input Nose R compensation amount (increment)		
D	Disabled	Compensation number (tool position, tool radius compensation)	M L L M	-
	Enabled	Auto tool length measurement, deceleration range d		
	Enabled	Data settings		
	Disabled	Sub-program storage device number (.D)		
E	Enabled	N number of threads per inch, precision thread lead		-
F	Enabled	Feedrate		-
	Enabled	Thread lead		
G	Enabled	Preparation function code		-
H	Disabled	Sequence number in sub-program	M	-
	Disabled	Compensation number (tool position, tool length compensation)		
I	Enabled	Circular center coordinates	M L	○
	Enabled	Vector component for tool radius compensation/nose R compensation		
	Enabled	Custom fixed cycle hole pitch		
	Enabled	1st cut amount for deep hole drilling cycle (2)		
J	Enabled	Circular center coordinates	M L	○
	Enabled	Vector component for tool radius compensation/nose R compensation		
	Enabled	Custom fixed cycle hole pitch, angle		
	Disabled	Dwell at deep hole drilling cycle (2) return point		

Address	Decimal point command	Application	M/L	Decimal point check
K	Enabled	Circular center coordinates		○
	Enabled	Vector component for tool radius compensation/nose R compensation		
	Disabled	Custom fixed cycle hole count	M	
	Disabled	Hole drilling cycle repeat count	L	
	Enabled	2nd cut amount for deep hole drilling cycle (2)	L	
L	Disabled	Fixed cycle repeat count	M	-
	Disabled	Program tool compensation input type selection (L2, L10, L11)	L	
	Disabled	Data setting selection (L70)	L	
	Disabled			
M	Disabled	Miscellaneous function code		-
N	Disabled	Sequence number		-
	Disabled	Data setting, data number	L	
O	Disabled	Program number		-
P	Enabled/Disabled	Dwell time (parameter)		-
	Disabled	Sub-program call program number		
	Disabled	Program tool compensation input compensation number		
	Disabled	Custom fixed cycle hole count	M	
	Disabled	Helical pitch count	M	
	Disabled	Surface speed radius	M	
	Enabled	Scaling factor	M	
	Disabled	High-speed mode type	M	
	Disabled	2nd reference position number	L	
	Disabled	Constant surface speed control axis number	L	
	Disabled	MRC finished shape start sequence number	L	
	Enabled	Cut-off cycle, shift amount/cut amount	L	
	Disabled	Compound thread cutting cycle cut count, chamfering, nose angle	L	
	Enabled	Compound thread cutting cycle thread height	L	
	Disabled	Data settings large division number	L	
Disabled	Restore sequence number fro sub-program	L		
Enabled	Coordinate position data	L		
Q	Enabled	Deep hole drilling cycle cut amount	M	-
	Enabled	Back boring shift amount	M	
	Enabled	Fine boring shift amount	M	
	Disabled	Spindle minimum clamp rotation speed	L	
	Disabled	MRC finished shape end sequence number	L	
	Enabled	Cut-off cycle cut amount/shift amount	L	
	Enabled	Compound thread cutting cycle minimum cut amount	L	
	Enabled	Compound thread cutting cycle 1st cut amount	L	
	Enabled	Cut amount each time for deep drilling cycle 1	L	
	Disabled	Program tool compensation input hypothetical nose point number	L	
	Disabled	Dwell at cut point for deep drilling cycle 2	L	

Address	Decimal point command	Application	M/L	Decimal point check
R	Enabled	R-designated circular radius		○
	Enabled	Corner rounding circular radius (.R)		
	Enabled	Program tool compensation input compensation amount		
	Enabled	Fixed cycle R point	M	
	Disabled	Synchronous tap/asynchronous tap change	M	
	Enabled	Auto tool length measurement deceleration area r	L	
	Enabled	MRC longitudinal, edge, and escape amount	L	
	Disabled	MRC molding division amount	L	
	Enabled	Cut-off cycle return amount	L	
	Enabled	Cut-off cycle escape amount	L	
	Enabled	Compound thread cutting cycle finishing allowance	L	
	Enabled	Compound thread cutting cycle, cutting cycle taper difference	L	
	Enabled	Distance to R point for hole drilling cycle, deep hole drilling cycle 2	L	
	Enabled	Coordinate position data	L	
S	Disabled	Spindle function code		-
	Disabled	Spindle maximum clamp rotation speed	L	
	Disabled	Constant surface speed control surface speed	L	
T	Disabled	Tool function code		-
U	Enabled	Coordinate position data		○
	Disabled	2nd Miscellaneous function code	L	Only for axis name
	Enabled	Program tool compensation input	L	
V	Enabled	Coordinate position data		○
	Disabled	2nd miscellaneous function code	L	Only for axis name
	Enabled	Program tool compensation input	L	
W	Enabled	Coordinate position data		○
	Disabled	2nd miscellaneous function code	L	Only for axis name
	Enabled	Program tool compensation input	L	
X	Enabled	Coordinate position data		○
	Enabled	Dwell time		Only for axis name
	Disabled	2nd miscellaneous function code	L	
	Enabled	Program tool compensation input	L	
Y	Enabled	Coordinate position data		○
	Disabled	2nd miscellaneous function code	L	Only for axis name
	Enabled	Program tool compensation input	L	
Z	Enabled	Coordinate position data		○
	Disabled	2nd miscellaneous function code	L	Only for axis name
	Enabled	Program tool compensation input	L	

Performing a consecutive input mistake check when creating a program

(1) Press the menu , input a new file name, and press the key. → Only an EOR program is created.

(2) Press the menu. → The menu is highlighted.

(3) Edit the program. → A warning is displayed at the moment the key is pressed.

```
Memory: /Program
File 100
Line 1 - 3
1 G28 XYZ;
2 G00 X1 ;
3
```

(4) Correct the input mistake and continue with program creation.

Performing a batch input mistake check for an existing program

(1) Change the operation to memory mode or MDI mode and perform a machining program operation search.

(2) Press the **Edit** menu.



Machining programs appear from the top.

(3) Press the **Miss warning** menu.



A warning displays the position where an input mistake occurred. The cursor moves from its current position to the first input mistake that was found.

```
Memory:/Program
File 348           Editing
Line 11 - 26
11 F1000 S3000 ;
12 M3 ;
13 N20 ;
14 G41 D3 ;
15 G3 Z-20. I20. P3. ;
16 G1 X90. Y100. F1000 ;
17 G91 X100. ;
18 Y100 ;
19 X-100 ;
20 Y-110 ;
21 N30 ;
22 G90 G0 Z5. ;
23 G40 ;
24 G91 G28 Z0 ;
25 M2 ;
```

(4) Press the **Next miss** menu.



The cursor moves to the next input mistake position.

```
Memory:/Program
File 348           Editing
Line 11 - 26
11 F1000 S3000 ;
12 M3 ;
13 N20 ;
14 G41 D3 ;
15 G3 Z-20. I20. P3. ;
16 G1 X90. Y100. F1000 ;
17 G91 X100. ;
18 Y100 ;
19 X-100 ;
20 Y-110 ;
21 N30 ;
22 G90 G0 Z5. ;
23 G40 ;
24 G91 G28 Z0 ;
25 M2 ;
```

(5) Correct the input mistake and continue with program creation.

Precautions

- (1) The input mistake check timing is outlined in the following table.

Timing	Check range	Check item	Example
When an edit key (alphabet, numbers, symbols, [Delete] etc.) is pressed (Does not include the cursor key and [Page Up/Down] keys.	1 block where the cursor is located	Decimal point	A warning is displayed upon pressing <input type="button" value="INPUT"/> when "X10;" is input.
			A warning is displayed upon pressing <input type="button" value="INPUT"/> when the "X10.Y" "." is inserted in the case where "X10Y" is input.

If the menu is pressed again when the input mistake check warning function is enabled, this function is then disabled.

The character color of the location of an input mistake then returns to normal.

- (2) The menu is grayed out and cannot be selected when the menu is not highlighted.
- (3) An input mistake check warning function can be performed even when editing a MDI program.
- (4) An input mistake check warning function can also be performed for a comment block.
- (5) If the data in an address that is subject to a decimal point input mistake is "0", no warning is issued regardless of the presence of a decimal point.
((Ex.)) No warning is issued for "X0"
 If the data in an address that is subject to a decimal point input mistake has been omitted, no warning is issued.
((Ex.)) No warning is issued for "G28XYZ")
- (6) Blocks with "[,]" are not subject to an input mistake check warning function.
- (7) The input mistake status is held in the operation screen edit window, setup screen MDI edit window, and edit screen edit area.
- (8) If an input mistake is found after the current cursor position, the "Input miss was detected" message appears. If no input mistake is found, the "Input miss was not detected" message appears.
- (9) With the multi-program display type, the input mistake check is performed also in the non-active area. Note that the messages "Input miss was detected" and "Input miss was not detected" will be displayed by judging the input mistake referring to the cursor position in the active area.
- (10) Input mistake check warning function is disabled in mass-editing.
- (11) "Executing" is displayed in flickering during checking input mistakes.

4.2.17 Adding Sequence No. (N No.) Automatically

When the sequence No. (N No.) is input once, the sequence No. to which a constant value is added can be automatically added to each block of the machining program.

The added sequence No. data is to N2 to N999999.

This function is disabled during mass-editing.

Operation method

- (1) Press the menu **N auto add**. → The menu key is highlighted and turned to the additional value input wait status. The cursor appears in the input area.

- (2) Input the additional value.
(Ex.) 10 **INPUT** → "N+10" is displayed in N No. additional value display column. The cursor moves from the input area to the edit area.

<When creating a new program>

- (3) Press the menu **Open (New)**, then press the menu **Memory**. → The list appears as a pop-up window. The machining programs in memory are displayed.

- (4) Input a file name not used, then press the **INPUT** key.
(Ex.) 100 → "O100" is created in edit area.

- (5) Input a program with N No.
(Ex.) N100G28XYZ →

```
Memory:/Program
File 101
Line 1 -      N+10  Editing
1 N100G28XYZ
```

<When editing an existing program>

- (3) Press the menu **Open**, then press the menu **Memory**. → The list appears as a pop-up window. The machining programs in memory are displayed.

- (4) Select a file, then press the **INPUT** key. → The selected file is opened in edit area.

- (5) Move the cursor to the position where data is to be inserted. Press the **INSERT** key. →

```
Memory:/Program
File 102
Line 1 -      N+10  Editing INS
1 N10 G28 XYZF1000 ;
2 N20 G00 Z-500. ;
3 N30 G00 X50. Y50. ;
4 M02 ;
5 %
```

- (6) Input " ; " (EOB). → "N110" (previous block N No. "N100" + an additional value "10") is added after a " ; " (EOB).

```
Memory:/Program
File 101
Line 1 -      N+10  Editing
1 N100G28XYZ
```

- (7) Input the second block data and " ; "
(EOB).
(Ex.) G92X0.Y0.Z0.;



"N120" is added behind " ; ".

```
Memory:/Program
File 101
Line 1 -          N+10  Editing
1 N100G28XYZ;N110G92X0.Y0.Z0.;N120
```

- (8) Press the **INPUT** key.



The message "Editing" disappears. The program is saved.

```
Memory:/Program
File 101
Line 1 -          N+10
1 N100 G28 XYZ;
2 N110 G92 X0. Y0. Z0. ;
3 N120 ;
4
```

The N No. data is automatically generated until the edit is interrupted with the **INPUT** key.

- (Note 1)** When the menu **N auto add** is pressed again while the menu is highlighting, N No. additional value display column is cleared, and N No. automatic addition is canceled.
- (Note 2)** The setting range is 1 to 1000. When "01" is input, "1" is set. When "0" is input, this function is canceled. When an illegal value or 9 characters or more is input, an error will occur.
- (Note 3)** The additional value can be set even when the machining program has been not opened. Even if the machining program is opened again, the additional value is not invalid.
- (Note 4)** This function is enabled only during regular editing of edit screen.
- (Note 5)** This function is disabled during mass-editing. However, the set N No. additional value in regular editing is not cleared, and the additional value is valid during regular editing again.
- (Note 6)** Even if the same N No. exists by automatic addition, an error will not occur.
- (Note 7)** The system separately has N No. in a right area and left area of the edit screen.
- (Note 8)** When the previous block is leading zero like "N010", N No. automatic addition can be executed. However, "0" is not added, and "N10" is displayed.

Example when N No. is added

- (1) When the previous block has N No. (When the cursor is located at the last character in the overwrite mode.)

N100 G28 X0 Y0 Z0 ; -> Input ";" -> N100 G28 X0 Y0 Z0;N110;
 N100 G28 X0 Y0 Z0 ; -> Input ";" -> N100 G28 X0 Y0 Z0 ;N110;
 N100 G28 X0 Y0 Z0 ; -> Input ";" -> N100 G28 X0 Y0 Z0 ;N110;

- (2) When the previous block has N No. (When the insertion mode)

N100 G28 X0 Y0 Z0 ; -> Input ";" -> N100 G2;N110 8 X0 Y0 Z0 ;
 N100G28X0Y0Z0;N200G90X20.Y20.Z20.;
 -> Input ";" -> N100G28X0Y0Z0;N110 N200G90X20.Y20.Z20.;

Example when N No. is not added

- (1) When the previous block has no N No.

G00X10.% -> Input ";" -> G00X10.;%

- (2) When the head of previous block has no N No.

Usually

G00N100X10.% -> Input ";" -> G00N100X10.;%

When comment

(N100)G00N100X10.% -> Input ";" -> (N100)G00N100X10.;%

When the previous block has a space

N100G00N100X10.% -> Input ";" -> N100G00N100X10.;%

- (3) When the N No. of previous block is over 9 characters.

N000001000% -> Input ";" -> N000001000.;%

- (4) When limits of total characters display per block is exceeded.

- (5) When the N No. is over 1000000.

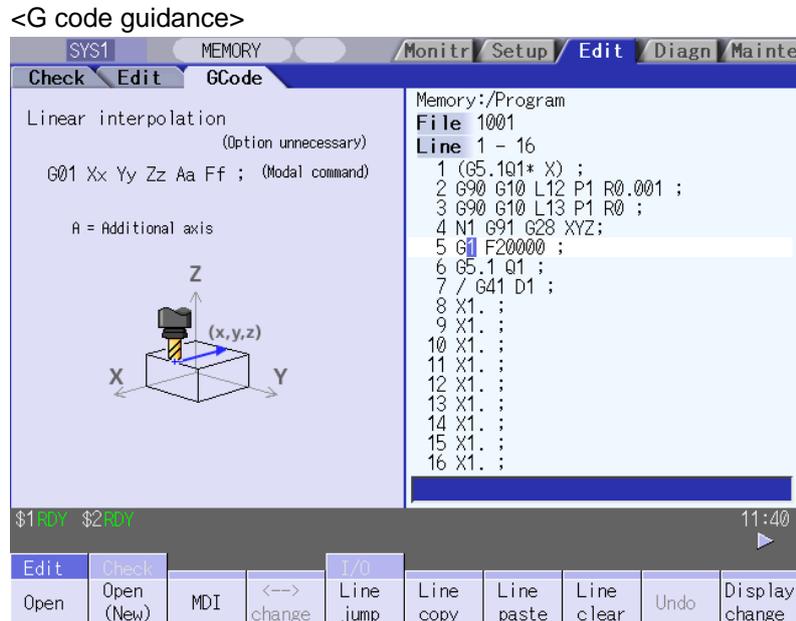
- (6) When the N No. automatic additional value is not set.

- (7) When the display character string in overwrite mode is interrupted.

N100 G28 X0 Y0 Z0 ; -> Input ";" -> N100 G28 X0;Y0 Z0 ;

4.2.18 G Code Guidance

G code guidance is a function that shows the command format details or the outline of its operation for the G code being edited while creating or editing the machining program. With this function, the G code format can be confirmed immediately. G code guidance is also displayed in the same manner even when MDI program is displayed.



The contents displayed on the G code guidance area depends on the cursor position on the Edit screen, and is refreshed when moving the cursor.

The G code that meets following conditions are displayed in this area.

- G code that exists between the head of block and cursor position including the cursor position.
- G code that exists immediately before the cursor if multiple G codes meet the above condition.

Specific example

G91 G17 ; █	➔	The guidance will not be refreshed because the cursor is out of the block.
G91 G17 █	➔	The guidance for G17 will be displayed.
G91█G17 ;	➔	The guidance for G91 will be displayed.
G91 G00 X100. █100. ;	➔	The guidance for G00 will be displayed because G00 is the nearest G code to the cursor.

G codes displayed in the G code guidance area

With G code guidance function, the commands having "G" followed by 1 or more numbers are regarded as the G code. Thus, when "G" that is located immediately before the cursor is within the comment, the guidance will not be refreshed. In the same manner, when a G code is commanded using variables such as "G#100" or "G#500", etc. but numbers, the guidance will not be refreshed.

The G code that can be displayed in the guidance area is as follows.

Gnnnnnnn.m L l ("n", "m", and "l" are arbitrary number.)

Commanded value	Number of valid digits	Remarks
n (integer)	1 to 8	When including "0" before integer command value, unnecessary "0" is ignored.
m (decimal places)	0, 1	
l (value commanded using "L")	0 to 2	This is valid when "L" exists immediately after the integer command value or decimal command value and is followed by a command value. If the command with "L" address precedes G code, the "L" command will not be described in the guidance.

(Note 1) As for the G codes not included in the specifications, the guidance will not be displayed even if the above condition is met.

G00 X100. (GCODE) ;	➔	"G" included in the comment "GCODE" is ignored, thus, G code guidance will not be refreshed.
N11 GOTO1 ;	➔	"G" included in the command "GOTO" is ignored, thus, G code guidance will not be refreshed.
G91 G120 X100 Y100. ;	➔	G code "G120" is not included in the specifications, thus, G code guidance will not be refreshed.
G0000031.1 ;	➔	The guidance for G31.1 will be displayed.
G0000031.10 ;	➔	Commanded value exceeds the number of valid digits, thus, G code guidance will not be refreshed.
L50 G10 ;	➔	The guidance not for "G10 L50" but "G10" will be displayed.

Precautions for G code guidance

- (1) G code guidance may be refreshed when the cursor is located near the comment.

[Ex.] `(PROG01)`; → The guidance for G01 will be displayed.

- (2) When the cursor is located at the axis address, etc., the guidance for the nearest G code will be displayed even if the address has no relationship with the G code.

[Ex.] `G01 G90 X100`; → G code that has a relationship with "X100." is "G01", however, G code guidance for "G90" will be displayed.

- (3) When moving the cursor to the block with no G code as follows, the contents of guidance will differ depending on the cursor's route even if the cursor is in the same block.

[Ex.]

<pre> N01 G91 G01 X100. F1000 ↓ N02 Y100. Z100. ← ... a ↑ b N03 G02 X50. Y50. R50. ; </pre>	<p>→ When the cursor is moved by the route "a", the guidance for G01 which is displayed last in N01 block will also be displayed in N02 block. When the cursor is moved by the route "b", the guidance for G02 which is displayed last in N03 block will also be displayed in N02 block.</p>
---	--

- (4) G code guidance will be refreshed even if an option of the G code is not valid. Thus, G code could not function though the guidance is displayed.
- (5) Basically, G code format in the guidance is displayed using the format of 3-axis specifications of X, Y, and Z.
- (6) Whichever plane is selected among the plane selection functions G17 to G19, the display data in the guidance is displayed using X-Y plane (G17) format.
- (7) When the G code command includes an address which have two or more meanings, both contents will be displayed.
- (8) G code guidance will not be refreshed while the character string is searched or replaced.
- (9) G code guidance cannot be displayed in mass-editing.

4.2.19 Playback Editing

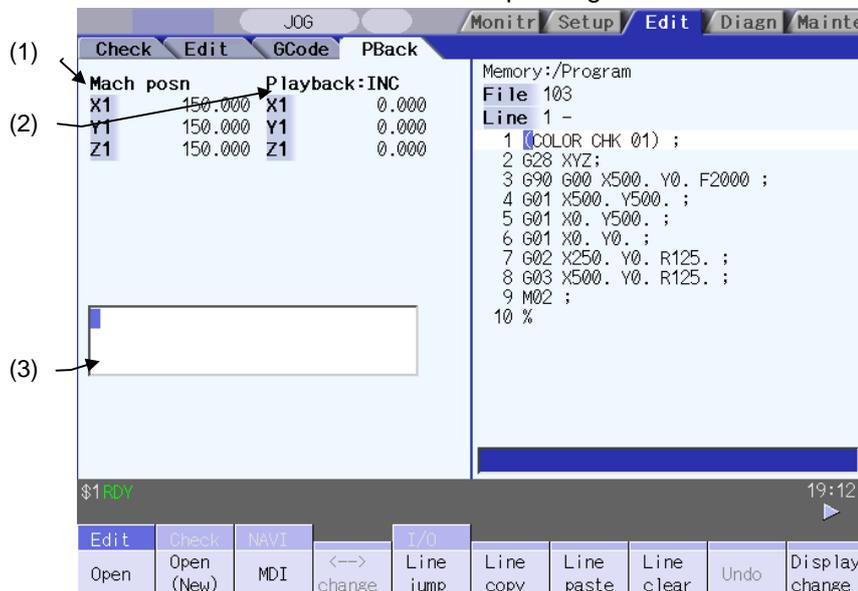
The playback function enables creation of a program while trying sample machining by manual (handle or jog) feed or mechanical handle feed.

A machining program can be created with movement distance data obtained by manual operation used as programmed command values.

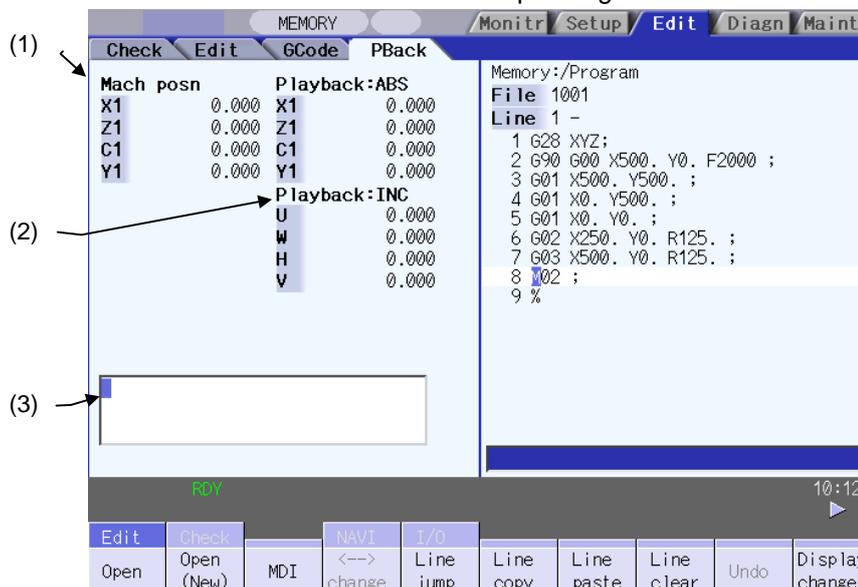
For L system, the playback counter and valid address key (X, Y, U, V, etc.) are different depending on the parameter "#1076 AbsInc" (ABS/INC address) setting.

	#1076 AbsInc (For L system only)	#1126 PB_G90	Reference
M system	(This setting is meaningless.)	0 (Incremental value)	Absolute/incremental command depending on G command
		1 (Absolute value)	
L system	0 (Absolute/incremental command depends on G command)	(This setting is meaningless.)	Absolute/incremental command depending on address code
	1 (Absolute/incremental command depends on address code)		

<Absolute/incremental command depending on G command>



<Absolute/incremental command depending on address code>



Display items

Display item	Details																					
(1) Machine position counter	<p>This displays the machine position counter. Up to max. number of axes in a part system can be displayed.</p> <table border="1" data-bbox="544 405 1437 488"> <tr> <td>M system</td> <td>Max. eight axes</td> </tr> <tr> <td>L system</td> <td>Max. five axes, the sixth axis or more is switched by the menu [Next axis].</td> </tr> </table> <p>When the rotary axis (except for all coordinate linear type), the display range is 0.000° to 359.999°.</p>	M system	Max. eight axes	L system	Max. five axes, the sixth axis or more is switched by the menu [Next axis].																	
M system	Max. eight axes																					
L system	Max. five axes, the sixth axis or more is switched by the menu [Next axis].																					
(2) Playback counter	<p>This displays the playback movement amount (axis movement value). When "absolute/incremental command depending on G command", the title and playback movement amount differs according to the parameter "#1126 PB_G90". When "Absolute/incremental command depending on address code", both titles are displayed.</p> <table border="1" data-bbox="544 689 1437 860"> <thead> <tr> <th>#1126 PB_G90</th> <th>Title</th> <th>Playback movement amount</th> </tr> </thead> <tbody> <tr> <td>0 (Incremental value)</td> <td>Playback: INC</td> <td>Playback movement amount before axis moves + axis movement amount</td> </tr> <tr> <td>1 (Absolute value)</td> <td>Playback: ABS</td> <td>The program position + manual interrupt amount</td> </tr> </tbody> </table> <p>Up to max. number of axes in a part system can be displayed.</p> <table border="1" data-bbox="544 891 1437 974"> <tr> <td>M system</td> <td>Max. eight axes</td> </tr> <tr> <td>L system</td> <td>Max. five axes, the sixth axis or more is switched by the menu [Next axis].</td> </tr> </table> <p>When the rotary axis (except for all coordinate linear type), the display range is following below.</p> <table border="1" data-bbox="544 1028 1437 1144"> <tr> <td rowspan="2">Rotary axis</td> <td>INC</td> <td>-359.999° to 359.999° (Rotary type short-cut invalid, workpiece coordinate linear type)</td> </tr> <tr> <td></td> <td>-180.000° to 179.999° (Rotary type short-cut valid)</td> </tr> <tr> <td></td> <td>ABS</td> <td>0.000° to 359.999°</td> </tr> </table>	#1126 PB_G90	Title	Playback movement amount	0 (Incremental value)	Playback: INC	Playback movement amount before axis moves + axis movement amount	1 (Absolute value)	Playback: ABS	The program position + manual interrupt amount	M system	Max. eight axes	L system	Max. five axes, the sixth axis or more is switched by the menu [Next axis].	Rotary axis	INC	-359.999° to 359.999° (Rotary type short-cut invalid, workpiece coordinate linear type)		-180.000° to 179.999° (Rotary type short-cut valid)		ABS	0.000° to 359.999°
#1126 PB_G90	Title	Playback movement amount																				
0 (Incremental value)	Playback: INC	Playback movement amount before axis moves + axis movement amount																				
1 (Absolute value)	Playback: ABS	The program position + manual interrupt amount																				
M system	Max. eight axes																					
L system	Max. five axes, the sixth axis or more is switched by the menu [Next axis].																					
Rotary axis	INC	-359.999° to 359.999° (Rotary type short-cut invalid, workpiece coordinate linear type)																				
		-180.000° to 179.999° (Rotary type short-cut valid)																				
	ABS	0.000° to 359.999°																				
(3) Playback editing area	<p>The contents of playback editing is input. When the axis address keys such as X and Y are input, the playback counter value currently displayed is input after the axis address.</p>																					

Operation method (Absolute/incremental command depending on G command)

(1) Set "#1126 PB_G90" to "0" on the parameter screen.



Incremental value mode is set.

(2) The menu **Display change** or **[→]** key is press several times on the edit screen.



The "playback display type" is displayed.
The menu **<--> change** cannot be used.

Mach	posn		P layback:INC
X1	150.000	X1	0.000
Y1	150.000	Y1	0.000
Z1	150.000	Z1	0.000

When the incremental value is set, the initial value is "0".

(3) Open the program pressing **Edit** and **Open** on the edit screen.



The selected program is displayed in right area.

Mach	posn		P layback:INC
X1	150.000	X1	0.000
Y1	150.000	Y1	0.000
Z1	150.000	Z1	0.000

File	103
Line	1 -
2	G28 XYZ;
3	G90 G00 X500. Y0. F2000 ;
4	G01 X500. Y500. ;
5	G01 X0. Y500. ;
6	G01 X0. Y0. ;
7	G02 X250. Y0. R125. ;
8	G03 X500. Y0. R125. ;
9	M02 ;
10	%

(4) Move the axis during manual mode.



The axis movement amount is displayed in playback movement amount.

Mach	posn		P layback:INC
X1	100.000	X1	-50.000
Y1	190.000	Y1	40.000
Z1	180.000	Z1	30.000

In the incremental value mode, the axis movement amount by manual mode is added to currently playback movement amount.

- (5) Input the data such as the G codes in the area for playback editing.



The alphanumeric characters, , , and **DELETE** keys can be input in the area for playback editing.

Mach	posn	Playback:INC
X1	100.000	X1 -50.000
Y1	190.000	Y1 40.000
Z1	180.000	Z1 30.000

F3000G01

- (6) Press the axis address key such as  or .



The playback movement amount is input after the axis address in the playback editing area. If the axis address key is pressed while the playback movement amount is being changed, the playback movement amount at the time the key was pressed will be input.

Mach	posn	Playback:INC
X1	100.000	X1 -50.000
Y1	190.000	Y1 40.000
Z1	180.000	Z1 30.000

F3000G01X-50.

(Note) When inputting a comment, etc., the characters that are the same as the axis address will all be handled as the axis address.

- (7) Press the **INPUT** key.



The program in the playback editing area will be inserted at the block before the cursor position in the right area, and the program is saved.

When there is no "EOB" (;) at the end of the character string, it is added automatically.

The cursor moves to the line head being cleared the content of the playback edit.

For an absolute value, the playback movement amount will not change.

For an incremental value, the difference of the axis command value for the axis used in step (6) and the playback movement amount will remain as the playback movement amount.

(Note) If the program is not displayed on right area, the operation message "Can't execute playback edit" appears.

(Note 1) When the program is not displayed on the right area, a playback editing cannot be executed.

(Note 2) When the **C.B** key is pressed, all data in the playback editing area will be erased.

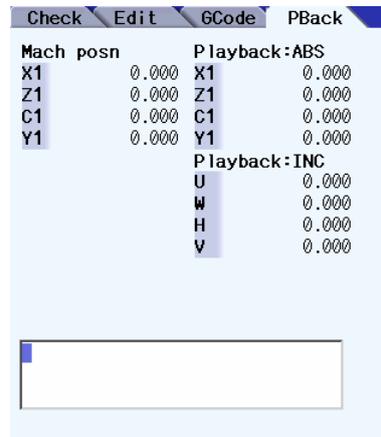
(Note 3) When the  and  keys are pressed, the program cursor on the right area will move.

(Note 4) When the page-switching key is pressed, the program on the right area will scroll.

(Note 5) When the menu key is pressed, the process corresponding to the right area will run.

Operation method (Absolute/incremental command depending on address code)

(1) Confirm that the parameter "#1076 AbsInc" is "1".



(2) Edit a program the same as the procedure (2) to (6) of "Absolute/incremental command depending on G command".

Notes for playback editing

- (1) Playback editing of a program in automatic operation or program restart is not possible.
- (2) Up to 95 characters can be edited in the playback editing area.
- (3) If the screen/part system is changed or the program file is opened when the input program is in the playback edit area, the program being edited will be aborted.
- (4) Other files can be opened during playback editing. Playback editing can be continued even when other file is opened. However, if an incremental value is designated, the playback movement amount will be cleared to 0 for all axes. A new file can be created and edited, and MDI editing carried out in the same manner.
- (5) If the program being playback edited automatically starts, playback editing cannot be continued.
- (6) The playback movement amount after playback editing will differ according to the parameter "#1126 PB_G90" state and the commanded G code. The playback movement amount after **INPUT** is pressed is calculated for each block that has been playback edited.

X axis playback movement amount before editing	Details to be playback edited	X axis playback movement amount after INPUT	
		For an absolute value	For an incremental value
X 100.000	G01 X10.;	X 100.000 The program position + manual interrupt amount is inserted as the playback movement amount.	X 90.000 The difference of the axis command value (X10.) and playback movement amount before INPUT (X100.) is inserted as the playback movement amount.
	G92 X10.;	X 10.000 The command value (X10.) following G92 is inserted as the playback movement amount.	X 0.000 Regardless of the command value following G92, 0 is inserted as the playback movement amount.

- (7) The playback editing is disabled during mass-editing.
- (8) Do not set the same address to the parameter "#1013 axname" and "#1014 incax". If the same address is set, the absolute counter has priority. (However, the same address as the address used in other part system can be specified.)
- (9) The playback editing cannot be enabled for an axis of auxiliary axis state. When an axis address of auxiliary axis state is input, a setting error will occur.

4.3 Program Check (2D)



The program check function (2D) is used to graphically draw the machining program movement path without performing auto operation.

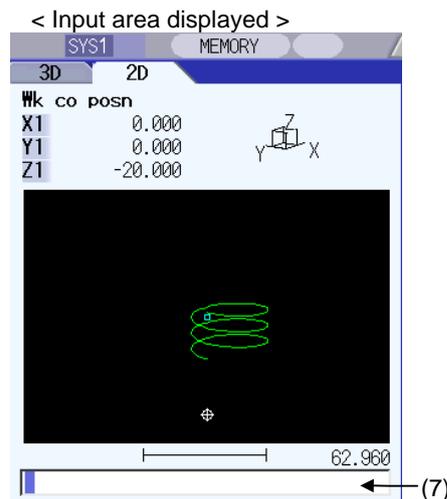
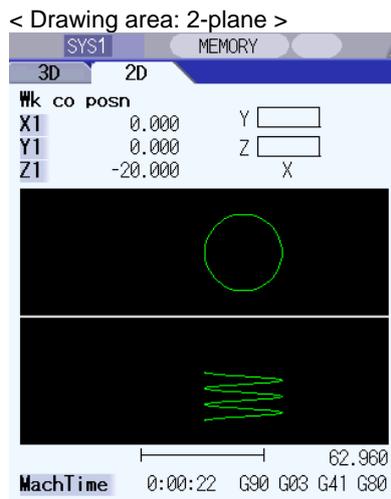
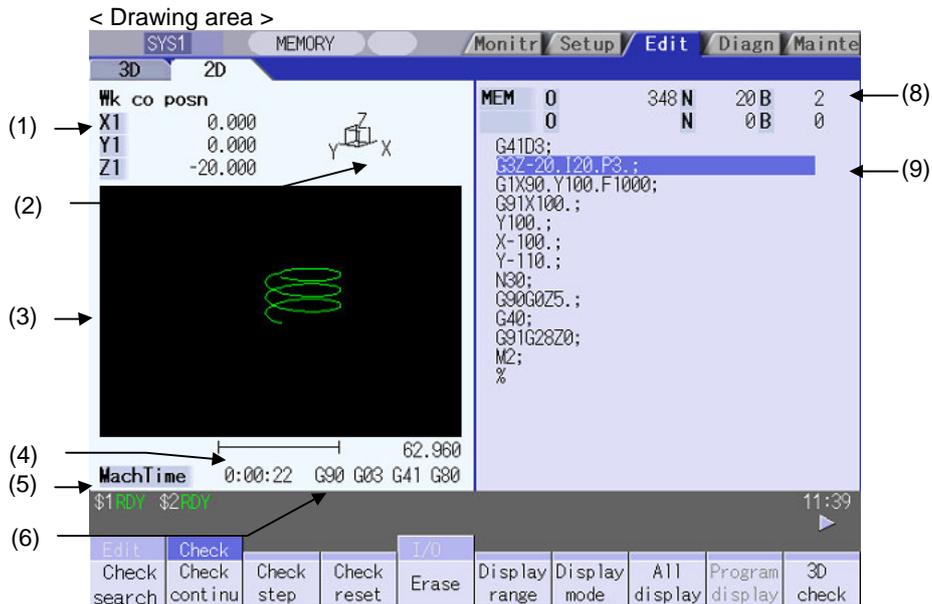
The machining program can be checked with graphic data drawn at a high speed.

Using All display menu, normal display and full-screen display modes can be switched.

(Note1) The program check (2D) function is an additional specification. The graphic check option is required.

(Note2) When changing the screen to the edit screen, display depends on the previous checking type (2D/3D). Note that when selecting this screen during graphic trace, 3D solid program check will apply. If the 3D solid check option is not valid in this time, the area of "Check" tag will not be blank.

■ Normal display (full-screen display mode is OFF)



Display item	Details
(8) Currently executed machining program	
Main O10...	This displays the device name, program number, sequence number, and block number currently being executed.
Sub O1234...	This displays the sub-program device name, program number, sequence number, and block number for the sub-program currently being accessed.
(9) Buffer display	This displays the contents of the machining program currently being checked. The block being executed is highlighted.

Menus

Menu	Details	Type	Reference
Check search	This performs a check search. Select a program to perform a check for from the program file list displayed in the pop-up window. For only the machining programs in NC memory, the check search is executed across all part systems in batch when the multi-part system program management is valid (#1285 ext21/bit0 =1). For 70 Series and FCU7-DA2-xx, this menu is "Operate search".	C	4.3.1 Checking Continuously
Check continu	This checks the program continuously.	B	
Check step	This checks the program per block.	B	4.3.2 Checking One Block at a Time
Check reset	This resets the program check.	C	4.3.3 Cancelling the Program Check
Erase	This erases the graphics displayed on the screen. Each time this menu is pressed, the graphic data is erased in the order of rapid traverse and cutting feed.	C	
Display range	This changes the graphic drawing display range. Press this menu to display the menu for changing the display mode. This changes the display range and erases the graphic data displayed on the screen.	C	4.3.5 Changing the Display Range
Display mode	This changes the drawing plane. Press this menu to display the menu for changing the display mode. There are 3 types of graphic drawing display mode: 1-plane, 2-plane, and 3-dimension. This changes the drawing plane and erases the graphic data displayed on the screen.	C	4.3.6 Changing the Display Mode
All display	This switches the normal display mode and the full-screen display mode.	B	4.3.8 Switching to Full-screen Display Mode
Program display	This displays the machining program being checked on the graphic drawing area. This menu can be selected only when the full-screen display mode is applied.	B	
3D check	This changes the program check (3D) display. (This menu does not display if the 3D solid graphic check option is disabled.) This menu cannot be selected while the check is performed or interrupted.	C	4.4 Program Check (3D)
Rotate	This sets the viewpoint angle for the 3-dimension display mode. This changes the viewpoint angle and erases the graphic data displayed on the screen. However, this menu cannot be used for other than the 3-dimension display mode.	C	4.3.7 Changing the Display Angle
Std range	The display range (scale and display position) is automatically set from the machine movable area. The machine movable area is set with the parameters "#2013 OT-" and "2014 OT+" (software limit). When the display range is changed, the graphic data displayed on the screen is erased.	C	

(Note) The menu **3D check** and **Operate search** keys cannot be selected while the check is performed or interrupted. Menu can be selected after **Check reset** is pressed.

4.3.1 Checking Continuously

Operation method

- (1) Press the main menu **Check**. The program check screen becomes active.
 Press the menu **2D Check** when the program check (3D) appears.

- (2) Press the **Check search** key. The following menu appears.
 The list appears in a pop-up window.

Memory	HD		Memory card	DS	FD	List update	Sort change		Close
--------	----	--	-------------	----	----	-------------	-------------	--	-------

(When specifications of all devices is valid.)

- (3) Select a device.
 (Ex.) Menu **Memory** The selected device name and directory (memory: /program) appear in the device name and directory display columns.

- (4) Use the , , , keys to align the cursor with the target machining program.
 The name of the machining program to be edited can be input in the input area.

(Note) When the number of characters of directory path (full path) exceeds 48, changing directories cannot be performed.

- (5) Press the **INPUT** key. The check search is started.
 A "Search complete" message appears when the search is completed.
 After performing the search, the device and program position appear in the current machining program display column.
 The list is then closed.
 The system distinguishes between the presence or lack of an extension, and therefore due care should be taken.

(Ex.)
 1001/1/2 O1001 N1 B2
 1001.PRG/1/2 O1001.PRG N1 B2
 /1/2 (current O number) N1 B2
 1001//2 O1001 N0 B2
 1001/1 O1001 N1 B0
 1001 O1001 N0 B0
 /1 (current O number) N1 B0
 //2 (current O number) N0 B2

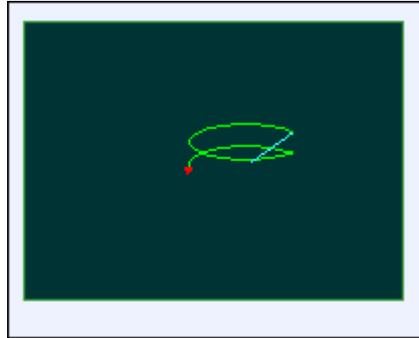
```
MEM 0 348 N 0 B 0
      0 N 0 B 0
-----
O348(PARTS1);
G28XYZ;
G92G53XYZ;
T3;
M6T5;
N10;
G90G0X0Y0;
Z5.;
GG43H3Z5.;
M8;
F1000S3000;
M3;
```

- (6) Press the key. 

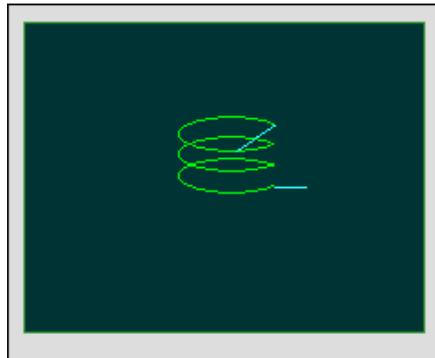
A block stoppage occurs when performing a program check if the or keys are pressed while performing a continuous check. Furthermore, the program check is restarted if the key is pressed during the block stoppage.

The program check is reset if the key is pressed while performing a continuous check.

A program check is executed and that path is drawn. A "Checking" message appears and the menu is highlighted. The check counter display is updated. The check G modal does not display. The machining time is updated.



A "Program check completed" message appears when the continuous check is completed. The menu highlight returns to normal.

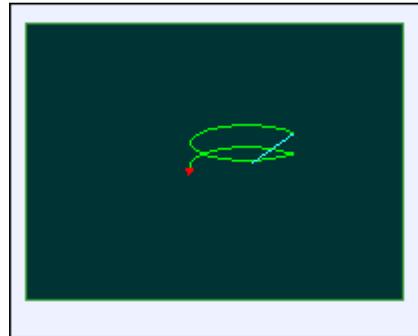


4.3.2 Checking One Block at a Time

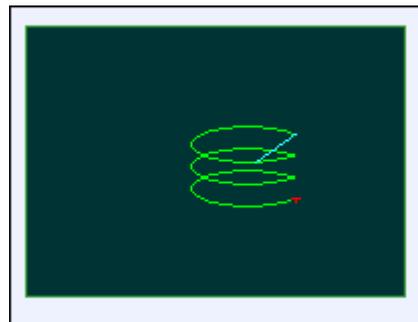
Operation method

- (1) Press the main menu .  The program check screen becomes active.
Press the menu key when the program check (3D) appears.
- (2) A check search is performed using the same method as for "4.3.1 Checking Continuously".
- (3) Press the menu .  The program check is executed for one block, and a graphic drawing of the machining program movement path is made.
The check counter display is updated.
The check G modal display is updated.
The machining time is updated.

<State during one block check>



<State when one block check is completed>

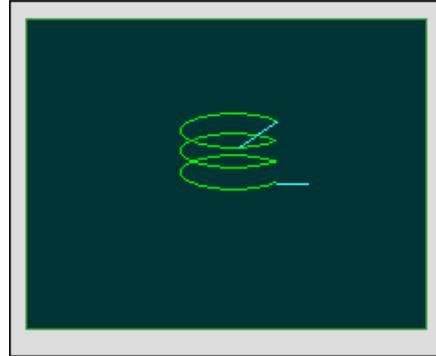


(4) Press the menu .



The program check is executed for the next block. The menu returns to normal state.

The program check is continued from the next block if the key is pressed during block stoppage.



4.3.3 Canceling the Program Check

Operation method

(1) Press the menu .



The program check is reset and a "Reset complete" message appears.

4.3.4 Drawing during a Program Check

During the program check, the machine position (tool center) path is drawn. However, if a radius compensation is applied on the program being checked, the program path and tool center are drawn. The colors of the lines displayed during the program check are expressed as follows.

		With radius compensation	Without radius compensation
Manual feed		-	-
Rapid traverse	Programmed path	White	-
	Tool center move path	Blue	Blue
Cutting feed	Programmed path	White	-
	Tool center move path	Green	Green

4.3.5 Changing the Display Range

The graphic drawing's scale can be enlarged or reduced, and the position moved or centered.

Operation method (Enlarging the and reducing the drawing)

- (1) Press the menu ➔ A white frame indicating the display range appears on the screen. The display range input mode is activated, and the following menu appears.

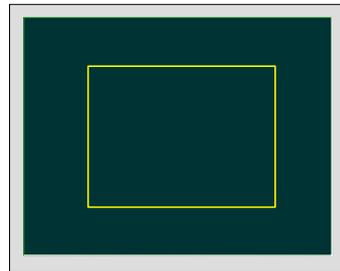
Zoom in	Zoom out	Up	Down	Left	Right	Centering			
---------	----------	----	------	------	-------	-----------	--	--	--

- (2) Press the menu or ➔ The size of the white frame can be changed by using key operations.

To enlarge the drawing size:
Press the menu or key.

To reduce the drawing size:
Press the menu or key.

A solid-line frame appears with respect to the original scale when enlarging, and a dotted-line frame appears when reducing.



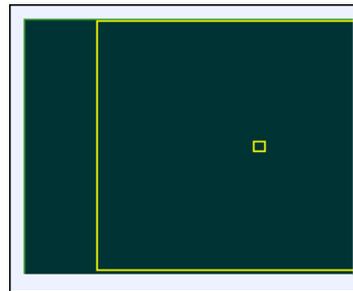
- (3) Press the key. ➔ The display scale changes. By changing the display scale, the graphic data displayed on the screen is erased.

Operation method (Changing the drawing display position)

- (1) Press the menu . ➔ A white frame indicating the display range appears on the screen. The display range input mode is activated, and the following menu appears.

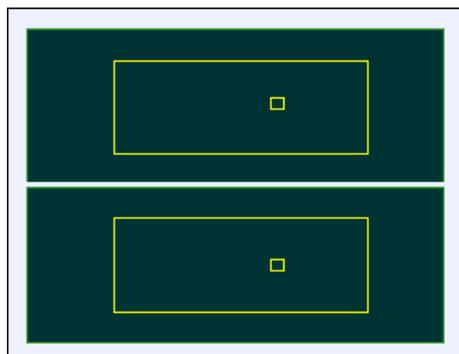
Zoom in	Zoom out	Up	Down	Left	Right	Centering			
---------	----------	----	------	------	-------	-----------	--	--	--

- (2) Press the , , , menu or cursor movement keys , , , . ➔ The cursor (□) indicating the center of the display and the frame line move up, down, left and right according to the key operations.



- (3) Press the key. ➔ The display position changes so that the cursor position (□) is the center of the area. By changing the display position, the graphic data displayed on the screen is erased, however, the scale value is not changed.

(Note 1) When changing the display range while the 2-plane display mode such as "XY/XZ", the display range (scale and display position) for the upper and lower areas changes in the same manner. The operation method is the same as the 1-plane display mode.



The white-framed display range in the upper and lower areas simultaneously move to the left/right when the , keys are pressed. The white-framed display range of either upper or lower areas moves up and down when the , keys are pressed. Which frame to be moved up/down can be changed with the page key.

(Note 2) The set displayed position will be held even after the power is turned OFF and ON again.

Operation method (Centering)

- (1) Press the menu .  A white frame indicating the display range appears on the screen.
The display range input mode is activated, and the following menu appears.

Zoom in	Zoom out	Up	Down	Left	Right	Center- ing			
------------	-------------	----	------	------	-------	----------------	--	--	--

- (2) Press the .  The display position is changed so that the current machine position appears in the center of the drawing area.

(Note 1) In the 2-plane display mode such as "XY/XZ", centering is applied for the upper and lower areas.
The operation method is the same as the 1-plane display mode.

(Note 2) When the menu key is pressed after the display scale has changed, only the centering is applied without changing the scale value.

4.3.6 Changing the Display Mode

There are three types of graphic drawing mode: 1-plane, 2-plane, and 3-dimension. Press the menu key, and by selecting one of the following display mode menus, the axis configuration for each plane changes, and the menu display returns to normal. It is also possible to change the display mode by setting the axis name in the input area.

- (Note 1) When the display mode is changed, the graphics displayed up to that point are erased.
- (Note 2) These set display mode is maintained even after rebooting the machine.
- (Note 3) The display mode can be set independently at the trace function and program check function. These settings are not linked with one another.
- (Note 4) The menu axis names XYZ are compatible with the basic axes IJK. X = base axis I, Y = base axis J, and Z = base axis K.

Display mode menu

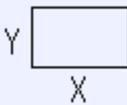
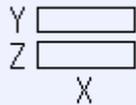
Menu	Details	Type
<input type="text" value="XY"/>	This changes to the 1- plane display mode configured of X-Y. The X-axis display in the horizontal axis, and the Y-axis displays in the vertical axis.	C
<input type="text" value="YZ"/>	This changes to the 1- plane display mode configured of Y-Z. The Y-axis display in the horizontal axis, and the Z-axis displays in the vertical axis.	C
<input type="text" value="XZ"/>	This changes to the 1- plane display mode configured of X-Z. The X-axis display in the horizontal axis, and the Z-axis displays in the vertical axis.	C
<input type="text" value="XY/XZ"/>	Changes to 2-plane display mode composing X-Y, and X-Z. The X-axis display in the horizontal axis, and the Y-axis and Z-axis display in the vertical axis.	C
<input type="text" value="YX/YZ"/>	Changes to 2-plane display mode composing Y-X, and Y-Z. The Y-axis display in the horizontal axis, and the X-axis and Z-axis display in the vertical axis.	C
<input type="text" value="XYZ"/>	This changes to the three-dimensional display mode. A cube is displayed on the lower right of the screen.	C

Changing the display axis name

- (1) Set the axis name and press the key.  XYZ appears the graphic display mode name. There are 3 drawing axes: X, Y, and C.
- XYC

There are three types of display mode: 1- plane, 2- plane, and 3D. An image displays which display mode is currently selected. The selected axis name displays for the image axis name.

Display example

<1-plane> (XY)	<2-plane> (XY/XZ)	<3-dimension> (XYZ)
		

4.3.7 Changing the Display Angle

This is used to set the graphic drawing display angle when in 3-dimension display mode.

Press the menu key or cursor keys to rotate the cubic displayed at the screen upper right and press the **INPUT** to set.

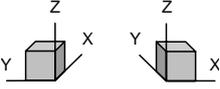
(Note 1) When the display angle is changed, the graphics displayed up to that point are erased.

(Note 2) The set display angle is maintained even after rebooting the machine.

(Note 3) The display angle can be set independently at the trace function and program check function. These settings are not linked with one another.

Operation method

- (1) Press the menu **Rotate**.  The mode changes to the rotation mode and the following menu appears.

- (2) Use the **Up**, **Down**, **Left**, **Right** menu to adjust the display angle.  Rotate the cubic display in the lower right of the screen using the operation keys.
 The cursor movement keys (, , , ) can also be used to rotate.

- (3) Press the **INPUT** key.  The display angle changes. By changing the display angle, the graphic data displayed on the screen is erased.

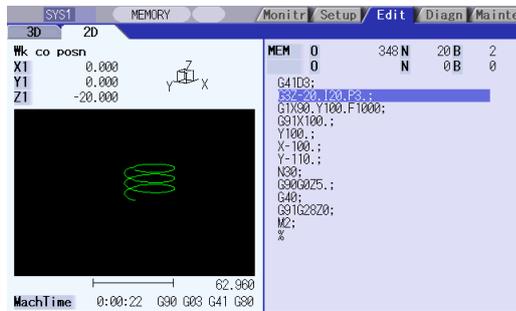
(Note 1) The set display angle is maintained even after rebooting the machine.

4.3.8 Switching to Full-screen Display Mode

Press the **All display** menu key to display the program check window on the whole screen. (full-screen display mode).

Press this menu again to return the normal display.

Switching the display mode

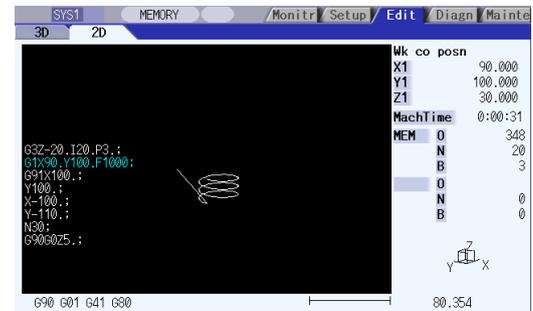


Normal display mode

All display
menu : ON



All display
menu : OFF



Full-screen display mode

(Note 1) The selected display mode is maintained even after rebooting the machine.

(Note 2) The full-screen display mode is applied to the trace function and program check (2D/3D solid) function commonly.

(Note 3) The full-screen display mode is applied commonly to all part systems.

Displaying a program

When pressing the **Program display** menu key, the contents of the machining program on checking is displayed on the drawing area.

When pressing the **Program display** again, the displayed program is erased.

Note that this menu can be used only when the full-screen display mode is selected.

(Note 1) The selected display mode is maintained even after rebooting the machine.

(Note 2) The program display mode is applied to the trace function and program check (2D) function commonly.

(Note 3) The **Program display** menu key cannot be selected while the normal display mode is selected. Program display mode is valid only when the full-screen display mode is selected.

(Note 4) The program display mode is applied commonly to all part systems.

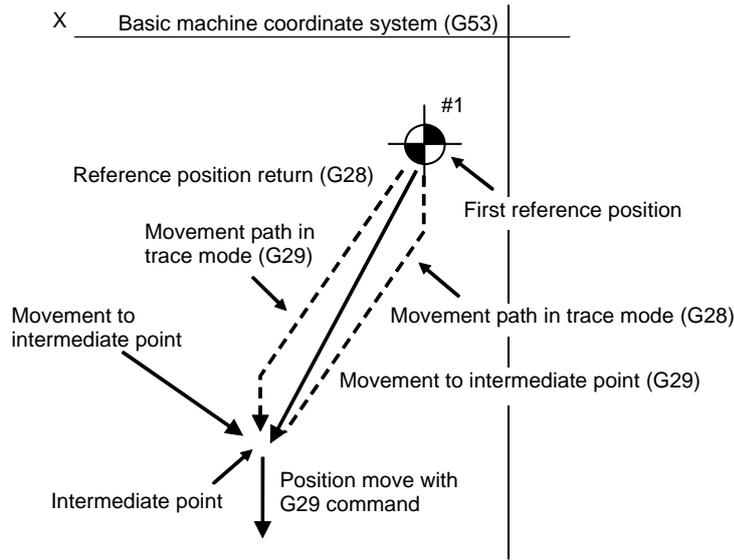
4.3.9 Availability of Check Mode with Other Functions

It is important to be aware that certain NC parameters, program commands, and machine operation panel switches cannot be used while performing a program check.

Function name	Remarks
Decimal point programming	Both decimal point input type I and II are valid.
Mirror image (Parameters, external input)	A mirror image is applied and a check drawing is made at the setting status prior to starting the check.
Z-axis command cancel	Signals are ignored and a check drawing is made.
Interlock	Signals are ignored and a check drawing is made.
External deceleration	Signals are ignored and a check drawing is made.
Override	Signals are ignored and a check drawing is made.
Auto operation pause (feed hold)	Signals are ignored and a check drawing is made.
Auto operation start-up (cycle start)	"M01 Operation alarm (0112)" occurs if automatic operation is started up while the check drawing is being made.
Auto-restart	The check is completed at M02/M30 and is not restarted.
Manual mode/handle	The mode can be changed to manual or handle mode and the machine can be moved even during the check. However, this is not reflected to the check drawing position.
F1-digit feed	The speed command does not influence the check drawing.
User macro I-II	Basic variable calculations, decisions and branch functions are all enabled. However, the following functions are ignored, or indefinite values are input. Macro interface input/output, Single block stoppage, Miscellaneous function complete signal waiting control, Feed hold, Feedrate override, G09 enabled/disabled control, Reading of position information for other than the previous block end point coordinates
M-S-T-2nd miscellaneous function commands	M-S-T-2nd miscellaneous function command signals commanded by the program during the check are not output. Neither do they display on the Operation screen etc.
G command modal	G modal displays on the modal display screen while the check drawing is being made.
Tool compensation amount	The check drawing is made with the tool compensation amount set when starting the check drawing.
Workpiece coordinate system offset amount	The check drawing is made with the workpiece coordinate system offset amount set when starting the check drawing. The display data is not changed when changes are made with G10 or system variables.
Parameter	The setting values when starting the check drawing are used. Parameters are changed to command values when changing the parameters using G10.
Optional block skip	The setting status when starting the check drawing are used.

(Note) Reference position return and start position return

The reference position return command and start position return command are valid, but actual machine motion (trace display) is partially different from the drawing by program check. As shown in the figure below, when returning to the reference position via the intermediate point with the G28 or G30 command, positioning to the intermediate point is done by a straight line and positioning from there to the reference position is done by each axis independently. With the G29 command, positioning to the intermediate point is done by each axis independently. Under program checking, however, positioning is always done by a straight line even when returning via the intermediate point; this may cause a little difference from the trace display.



4.3.10 Handling of Variable Commands, Parameter Input by Program, and Compensation Data Input by Program

If a program check is performed for a machining program for which the variable command data etc. is rewritten, all data types are treated as shown in the following table after completion.

Data type	Saving data before program check	Explanation
Parameter	×	The program parameter input commands are set as actual data. (The data before the check is not saved.)
Local variable	○ / ×	It is possible to return to the status prior to starting the program check based on the "#1231 set03(bit0) " graphic check interchangeable parameter value. 0: Returns to original status after check completion 1: Does not return to original status after check completion
Common variable	○ / ×	
Workpiece coordinate system offset	○ / ×	
Tool compensation amount	○ / ×	

4.3.11 Notes for Program Check (2D)

Check function

- (1) The check function draws while referring only to the NC internal operation results. Thus, the path is not drawn for commands requiring machine movement.
- (2) Check search and check start are not possible during automatic start or automatic pause. The operation message "Executing automatic operation" appears.
- (3) Operation search is possible for a program with 32 or less file name characters.
- (4) Graphic check is not available for MDI program.

Notes for checking (Common for continuous check and check step)

- (1) When the menu , , , or key is pressed while checking, the graphic data is erased, however, the program check is not interrupted.
- (2) Drawing is continued even after changing to another screen while checking.
- (3) Automatic operation cannot be performed even if the automatic operation button is pressed while checking. "M01 Operation Alarm (0112)" occurs.
- (4) Execute automatic start after the check is completed or after the check is interrupted by pressing the .
- (5) Refer to "4.3.10 Handling of Variable Commands, Parameter Input by Program, and Compensation Data Input by Program " for further details on all types of data set in the program while checking.
- (6) Checking the program stops if there is a M00/M01 command. However, when executing the M01 command, drawing is stopped regardless of the ON/OFF status of the optional stop switch signal.
- (7) When switching part systems during checking, drawing will be erased. When the displayed part system is undone, check operation will be continued from the previous state (program position before part systems were switched).
- (8) Status during checking (in "Check continu"/in "Check step") is held per part system.
- (9) 2D graphic check drawing begins from the actual coordinate value used for the program operation (coordinate value of the machine position).
- (10) The contents of ONB display and buffer display in the check screen during automatic operation are the contents of the program during automatic operation. The contents of ONB display and buffer display in the operation screen during check execution are the contents of the program during check execution.
- (11) When operation search or check operation ("Check continu" or "Check step") is performed during program restarting, the operation message "in program restarting" is displayed, and the program will not be searched or graphically checked.

Notes for ending check

- (1) If an M02/M30 command is found, "Program check completed" occurs.
- (2) By performing commands such as a check reset or turning the trace display ON, the check mode is exited. Clear the errors with these operations in the case of a program error. When the NC has been reset while the check screen is displayed, graphic check will be completed, but the menu for changing from 3D to 2D graphic check will remain disabled. Press the check reset menu to enable the switching operation.
- (3) Check reset is available only for the displayed part systems.
- (4) When a NC reset is performed during check operation, all the part systems will be reset. When a part system not being displayed is in checking and if switching to that part system, check will be executed from the head of the program.

Notes for switching between trace mode and check mode

- (1) The checking mode is released when the trace mode becomes ON.
- (2) Graphic check (2D) cannot be performed during the trace mode.
- (3) When displaying Edit screen during the graphic trace mode, graphic check 3D will be applied regardless of the previous check type. If the 3D solid check option is not valid, the area of Check tag will be blank.
- (4) When trace window is displayed during check, check will be terminated. (This applies for all the part systems under checking.)

4.4 Program Check (3D) [700 series only]



Program check (3D) is a function that draws the workpiece shape and tool movement in the cutting process of the machining program as a solid image without executing automatic operation. The machining program can be checked with graphic data drawn at a high speed.

This function requires workpiece and tool shape settings.

Program check (3D) can draw a solid image in the basic machine coordinate system or the workpiece coordinate system.

The coordinate system used to draw can be switched on "Work set" screen.

In the workpiece coordinate system, a solid image is drawn by using the workpiece coordinate system designated by the program.

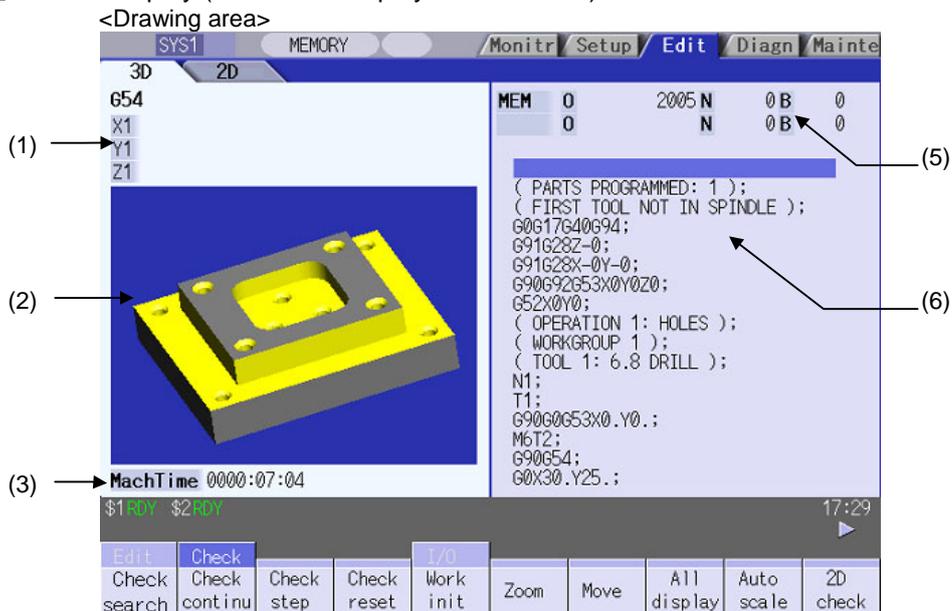
Using **All display** menu, normal display and full-screen display modes can be switched.

(Note 1) The program check (3D) function is an additional specification. The graphic check and 3D solid graphic check option is required.

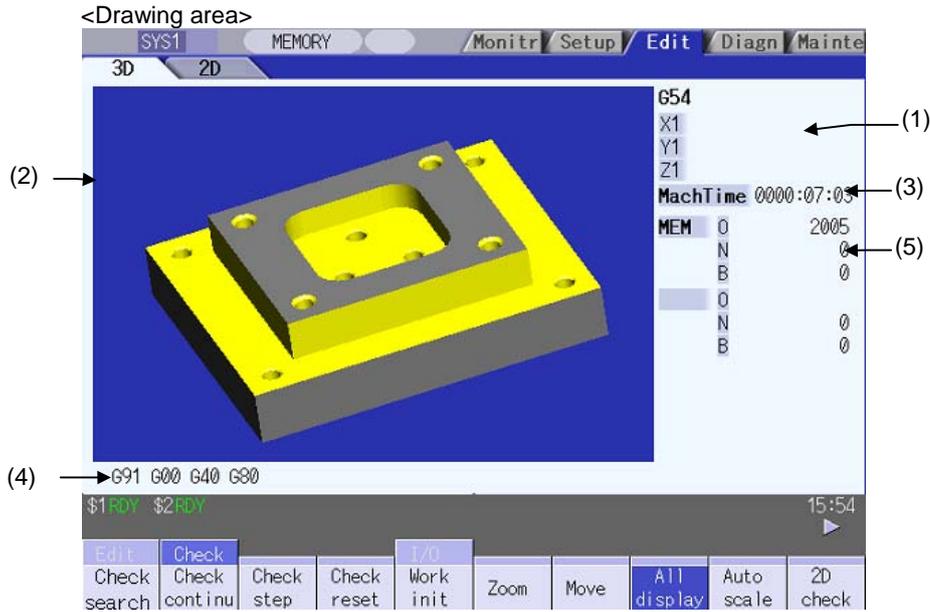
(Note 2) When changing the screen to the edit screen, display depends on the previous checking type (2D/3D). Note that when selecting this screen during graphic trace, 3D solid program check will apply.

If the 3D solid check option is not valid in this time, the area of Check tag will not be blank.

■ Normal display (full-screen display mode is OFF)



■ Full-screen display (full-screen display mode is ON)



Display items

Display item	Details
(1) Check counter	This displays the axis counter for check drawing. As for the counter, the machine position or workpiece coordinate position is displayed according to setting the drawing coordinate system on "Work set" screen. As for the axes name, system common axis name ("#1022 axname2") based on the basic axis name set to "#1026 base_I", "#1027 base_J", and "#1028 base_K" are displayed. (Note) When the setting values of #1026 to #1028 have been changed, the changed values will be reflected in the check counter display when the power is turned ON again.
(2) 3D drawing area	This area is used to display a solid graphic drawing of the workpiece shape and tool movement during the cutting process in the machining program. The drawing viewpoint can be moved, and the drawing size can be enlarged and reduced. The coordinate system used to draw switches the basic machine coordinate system or a workpiece coordinate system according to setting the drawing coordinate system on "Work set" screen.
(3) Machining time display	This calculates and displays the time taken for machining.
(4) Check G modal display	This displays the following modals: <ul style="list-style-type: none"> • Absolute/incremental (G90, G91: Group 3) • Operation mode (G00, G01, G02, G03 etc.: Group 1) • Tool radius compensation (G40, G41, G42: Group 7) • Fixed cycle (G80, G81 etc.: Group 9)
(5) Currently executed machining program	
Main O10...	This displays the device name, program No., sequence No., and block No. currently being executed.
Sub O1234...	When sub-program is being executed, the sub-program device name, program No., sequence No., and block No. are displayed.
(6) Buffer display	This displays the contents of the machining program currently being executed. The block being executed is highlighted.

Menus

Menu	Details	Type	Reference
Check search	This performs a check search. Select a program to be checked from the program file list displayed in the pop-up window. For only the machining programs in NC memory, the check search is executed across all part systems in batch when the multi-part system program management is valid (#1285 ext21/bit0 =1, ext21/bit1 =1). For 70 Series and FCU7-DA2-xx, this menu is "Operate search".	C	4.4.1 Checking Continuously
Check continu	This checks a program continuously.	B	
Check step	This checks a program per block.	B	4.4.2 Checking One Block at a Time
Check reset	This resets a program check.	C	4.4.3 Canceling the Program Check
Work init	This initializes the workpiece shape. Displays the pre-machining workpiece shape set at the "Work set" window (workpiece shape setting). At this time, the cross-section status is cancelled.	C	
Zoom	This enlarges/reduces the displayed workpiece shape.	C	4.4.4 Enlarging and Reducing the Workpiece Shape
Move	This moves the displayed workpiece shape.	C	4.4.5 Moving the Workpiece Shape
All display	This switches the normal display mode and the full-screen display mode.	C	
Rotate	This rotates the displayed workpiece shape.	C	4.4.6 Rotating the Workpiece Shape
Auto scale	The scale is automatically set so that the pre-machining workpiece shape set at the "Work set" window (workpiece shape setting) takes up approximately 90% of the drawing area.	C	
2D check	This changes the program check (2D) display. This menu cannot be selected while the check is performed or interrupted.	C	4.3 Program Check (2D)
Colli check	This enables or disables the interference check. When the interference check is enabled, if the tool and workpiece contact each other when performing rapid traverse (G0) movement, the contact location is drawn in a color highlighting the interference.	B	4.4.7 Performing an Interference Check
Work setting	This sets the workpiece shape used at the solid display.	C	4.4.9 Setting the Workpiece Shape
Tool setting	This sets the tool shape used at the solid display.	C	4.4.10 Setting the Tool Shape

(Note 1) When the graphic trace mode is entered and automatic operation is running, the menu **Operate search** , **Check continu** , **Check step** , **Check reset** cannot be selected and the 3D solid program check cannot be performed.

Only the 3D solid program check can be performed unless the automatic operation is running.

(Note 2) The menu **2D check** , **Operate search** , **Work setting** , **Tool setting** cannot be selected while the check is performed or interrupted.

4.4.1 Checking Continuously

Operation method

- (1) Press the main menu Check. ➔ The program check screen becomes active.
 Press the menu 3D Check key when the program check (2D) appears.

- (2) Press the Check search key. ➔ The following menu appears.
 The list appears in a pop-up window.

Memory	HD		Memory card	DS	FD	List update	Sort change		Close
--------	----	--	-------------	----	----	-------------	-------------	--	-------

(When specifications of all devices is valid.)

- (3) Select a device.
(Ex.) Menu Memory ➔ The selected device name and directory (memory: /program) appear in the device name and directory display columns.

- (4) Use the ↑, ↓, ↶, ↷ keys to align the cursor with the target machining program.
 The name of the machining program to be edited can be input in the input area.

(Note) When the number of characters of directory path (full path) exceeds 48, changing directories cannot be performed.

- (5) Press the INPUT key. ➔ The check search is started.
 A "Search completed" message appears when the search is completed.
 After performing the search, the function and program position appear in the current machining program display column.
 The list is then closed.
 The system distinguishes between the presence or lack of an extension, and therefore due care should be taken.

(Ex.)
 1001/1/2 O1001 N1 B2
 1001.PRG/1/2 O1001 N1 B2
 /1/2 (current O number) N1 B2
 1001//2 O1001 N0 B2
 1001/1 O1001 N1 B0
 1001 O1001 N0 B0
 /1 (current O number) N1 B0
 //2 (current O number) N0 B2

```

MEM 0          348 N    0B  0
    0          N    0B  0

O348(PARTS1);
G28XYZ;
G92G53XYZ;
T3;
M6T5;
N10;
G90G0X0Y0;
Z5.;
GG43H3Z5.;
M8;
F1000S3000;
M3;
    
```

- (6) Press the key. 
- A block stoppage occurs when performing a program check if the or keys are pressed while performing a continuous check. Furthermore, the program check is restarted if the key is pressed during the block stoppage.
- The program check is reset if the key is pressed while performing a continuous check.
- A program check is performed, and a solid drawing is made for the cutting process workpiece shape and tool movement.
A "Checking" message appears and the menu is highlighted.
The check counter display is updated.
The check G modal does not appear.
The machining time is updated.
- A "Program check completed" message appears when the continuous check is completed.
The menu highlight returns to normal.

4.4.2 Checking One Block at a Time

Operation method

- (1) Press the main menu . 
- Press the menu key when the program check (2D) appears.
- (2) A check search is performed using the same method as for "4.4.1 Checking Continuously".
- (3) Press the menu . 
- The program check is executed for one block, and a solid drawing of the workpiece shape and tool movement during the cutting process is made.
The check counter display is updated.
The check G modal display is updated.
The machining time is updated.
- (4) Press the menu . 
- The program check is continued from the next block if the key is pressed during block stoppage.

4.4.3 Canceling the Program Check

Operation method

- (1) Press the menu . 
- The program check is reset and a "Reset complete" message appears.

4.4.4 Enlarging and Reducing the Workpiece Shape

Operation method

- (1) Press the menu . ➔ The mode changes to the enlargement/reduction mode, and the following menu appears.
-
- (2) Press the menu . ➔ The drawing area center position is set in the center, and the enlarged workpiece is redisplayed.
- To enlarge the drawing:
Press the menu or key.
- To reduce the drawing size:
Press the menu or key.

4.4.5 Moving the Workpiece Shape

Operation method

- (1) Press the menu . ➔ The mode changes to the movement mode and the following menu appears.
-
- (2) Use the , , , keys to adjust the display position. ➔ The workpiece shape moved in the designated direction is redisplayed.
- The cursor movement keys (, , ,) can also be used for moving.

4.4.6 Rotating the Workpiece Shape

Operation method

- (1) Press the menu . ➔ The mode changes to rotation mode and the following menu appears.
-
- (2) Use the , , , keys to adjust the display angle. ➔ The workpiece shape rotated in the designated direction is redisplayed.
- The cursor movement keys (, , ,) can also be used to rotate.

(Note 1) The set display angle is maintained even after rebooting the machine.

4.4.7 Performing an Interference Check

If the tool and workpiece contact when performing rapid traverse (G0) movement, the contact location is drawn in a color highlighting the interference.

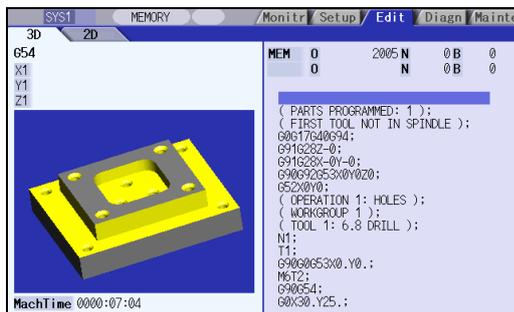
Operation method

- (1) Press the menu . → The menu is highlighted.
- (2) Press the menu or . → A program check is performed, and a solid drawing of the workpiece shape and tool movement during the cutting process is performed. A "Checking" message is displayed and the menu or key is highlighted. The workpiece and tool contact, this is drawn in a color highlighting the interference.

4.4.8 Switching to Full-screen Display Mode

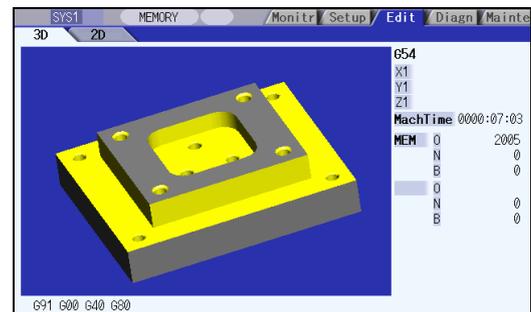
Press the menu key to enlarge the program check window (full-screen display mode). Press this menu again to return the normal display.

Switching the display mode



menu : ON
→
←

menu : OFF



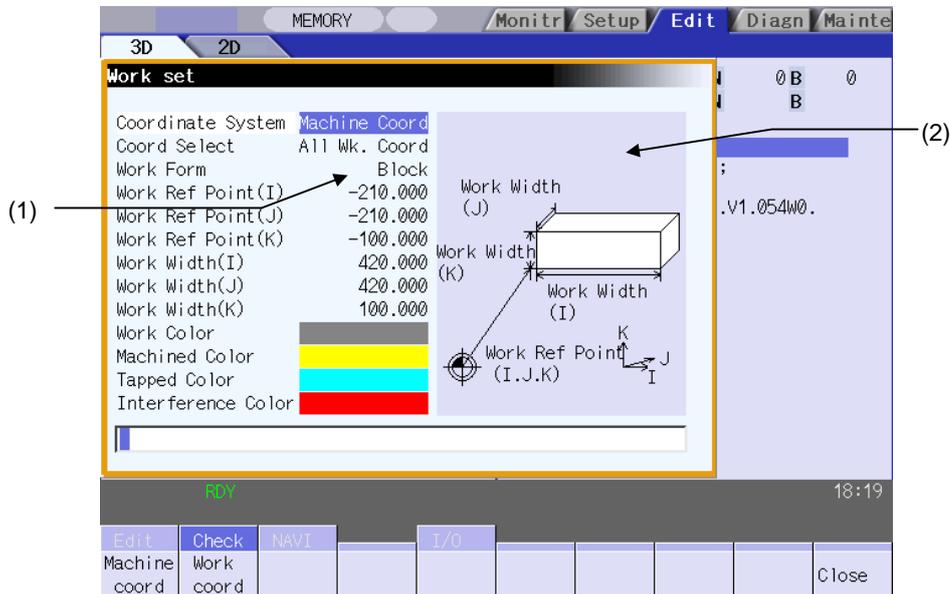
(Note 1) The selected display mode is maintained even after rebooting the machine.

(Note 2) The full-screen display mode is applied to the trace function and program check (2D/3D solid) function commonly.

(Note 3) The full-screen display mode is applied commonly to all part systems.

4.4.9 Setting the Workpiece Shape

Press the menu **Work setting** key to display a pop-up window similar to the following, and set the workpiece shape used in the solid display.



Display items

Display item	Details
(1) Workpiece shape setting area	This sets all workpiece shape items.
(2) Guide drawing	This displays a material shape guide drawing. The displayed guide drawing is changed according to the coordinates set on "Work set" screen.

Menus (the cursor is at CoordinateSystem (drawing coordinate system))

Menu	Details	Type	Reference
Machine coord	This specifies the coordinate system used to 3D drawing. The name and value of the check counter displayed on 3D check screen are switched to the machine position and the workpiece coordinates counter position. The default setting is the "Machine coord" (machine position). When "Machine coord" is selected, "All Wk. Coord" (all workpiece coordinates) is set at "Coord Select" (workpiece coordinate system selection).	C	Setting the drawing coordinate system (CoordinateSystem)
Work coord			
Close	This closes the pop-up window and quits this function.	C	

Menus (the cursor is at WorlForm (material shape))

Menu	Details	Type	Reference
Default shape	This sets the material shape. The drawings are made as a block when the default shape is selected.	C	"Setting the material shape"
Block			
Close	This closes the pop-up window and quits this function.	C	

Menus (the cursor is at Coord Select (workpiece coordinate system selection))

Menu	Details	Type	Reference
All wk. coord	This draws all workpiece coordinate systems. When the menu is selected, "All Wk. Coord" (all workpiece coordinates) is displayed at the cursor position. When the drawing coordinate system is "Machine coord" (machine coordinate), "All Wk. Coord" is selected. The default setting is "All Wk. Coord".	C	Selecting the drawn workpiece coordinate system
Coord G54-G59	This specifies the workpiece coordinate (G54-G59) to the drawn coordinate system. The drawn coordinate system is selected from sub-menu (G54 to G59). If the drawing coordinate system is "Machine coord", this is displayed in gray and cannot be selected.	C	
Coord G54.1P	This specifies the external workpiece coordinate (G54.1 Pn) to the drawn coordinate system type. By inputting P No., the external workpiece coordinate (G54.1 Pn) can be selected to the drawn coordinate system. If the drawing coordinate system is "Machine coord", this is displayed in gray and cannot be selected.	A	
Close	This closes the pop-up window and quits this function.	C	

Menus (the cursor is at WorkRefPoint(I,J,K) (material basic point), or WorkWidth (material width))

Menu	Details	Type	Reference
Close	This closes the pop-up window and quits this function.	C	

Menus (the cursor is at the WorkColor, MachinedColor, TappedColor, InterferenceColor, SectionColor in the window)

Menu	Details	Type	Reference
Default color	This sets the material color (WorkColor), machining surface color (MachinedColor), tap surface color (TappedColor), interference surface color (InterferenceColor), and cross section color (SectionColor). The drawings are made using the following colors when the default colors are selected. Material color : Gray Machining surface color : Yellow Tap machining surface color : Light blue Interference surface color : Red Cross section color : Green (Note) The tap surface color (TappedColor) of the work shape setting (Work set) screen is applied when the selected tool is tap. Therefore, switching of the tap surface color and normal surface color depends on not G command but the selected tool. However, even if the selected tool is tap, the tap surface color (TappedColor) of the work shape setting (Work set) screen is not applied when circular interpolation or helical interpolation is commanded.	C	"Setting the material color"
Gray			
Red			
Yellow			
Blue			
Green			
Light blue			
Purple			
Pink			
Close	This closes the pop-up window and quits this function.	C	

Setting the drawing coordinate system (CoordinateSystem)

(1) Use the , keys to move the cursor to the drawing coordinate settings column.

(2) Press the menu .

It is also possible to set input values.

0 : Machine coord

1 : Work coord



"Work coord" appears at the cursor position.
The cursor moves one place down.
A guide drawing with workpiece coordinate origin appears in the guide drawing area.

Selecting the drawn workpiece coordinate system

When the drawing coordinate system is the workpiece coordinate, the drawn coordinate system can be selected. Only the coordinate system selected here is drawn.

When the drawing coordinate system is the machine coordinate, "All Wk. Coord" (all workpiece coordinates) is set at "Coord Select" (workpiece coordinate system selection). Therefore, or menu cannot be selected.

(Example) When "Coord G54-G59" is selected

(1) Use the , keys to move the cursor to "Coord Select" column.

(2) Press the menu .



The sub-menus, which select G54 to G59, are displayed.

(3) Press the menu .



"G56" appears at the cursor position.
The cursor moves one place down.
The sub-menus of the material shape (WorkForm) are displayed.

When menu is selected, return to menu.

Setting the material shape (WorkForm)

(1) Use the , keys to move the cursor to the material shape settings column.

(2) Press the menu .



"Block" appears at the cursor position.
The cursor moves one place down.
A block guide drawing appears in the guide drawing area.

It is also possible to set input values.

0 : Default shape

1 : Block

Setting the material basic point (WokRefPoint (I)) and size (WrkWidth (I,J,K))

Enter the material basic point and size. Specify the material basic point with the specified coordinate on "Work set" screen.

(1) Use the , keys to move the cursor to the material basic point or width (I) setting column.

(2) Input data.
(Ex.) 100



"100.000" appears at the cursor position.
The cursor moves one place down.

(3) Other basic points and width data are set in the same way.

(Note 1) The workpiece will not be displayed if two or more size data are "0".

(Note 2) The setting range of the material basic point and width is as follows.
However, the value of less than 0 cannot be set to width.

#1003 iunit	#1041 I_inch			
	0 (mm)		1 (inch)	
B	-99999.999	to 99999.999	-9999.9999	to 9999.9999
C	-99999.9999	to 99999.9999	-9999.99999	to 9999.99999
D	-99999.99999	to 99999.99999	-9999.999999	to 9999.999999
E	-99999.999999	to 99999.999999	-9999.9999999	to 9999.9999999

Setting the material color (WorkColor)

(1) Use the , keys to move the cursor to the material color setting column.

(2) Press the material color menu.
(Ex.) Menu



The cell where the cursor is located appears in gray.
The cursor moves one place down. (The cursor does not move when at the cross section color.)

It is also possible to set input values.

- 0 : Default color
- 1 : Gray
- 2 : Red
- 3 : Yellow
- 4 : Blue
- 5 : Green
- 6 : Light blue
- 7 : Purple
- 8 : Pink

(3) Other color data is set in the same way.

(Note) The following colors are used if the default colors are designated.

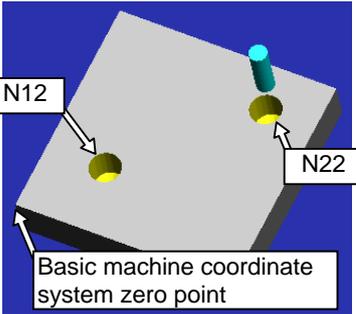
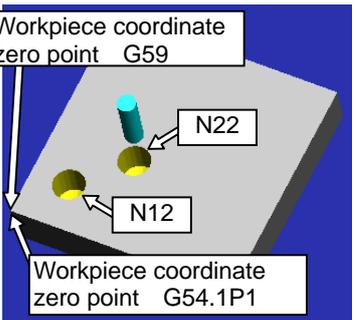
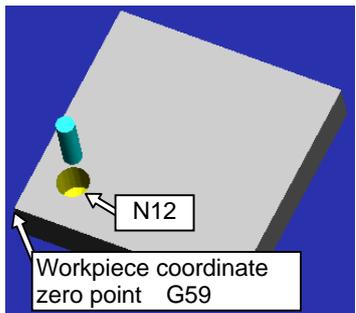
- Material color : Gray
- Machining surface color : Yellow
- Tap machining surface color: Light blue
- Interference surface color : Red
- Cross section color : Green

Difference of drawing by combination of "CoordinateSystem" and "Coord Select"

By combination of the drawing coordinate system (CoordinateSystem) and the workpiece coordinate system selection (Coord Select), the drawing at 3D check changes as follows.

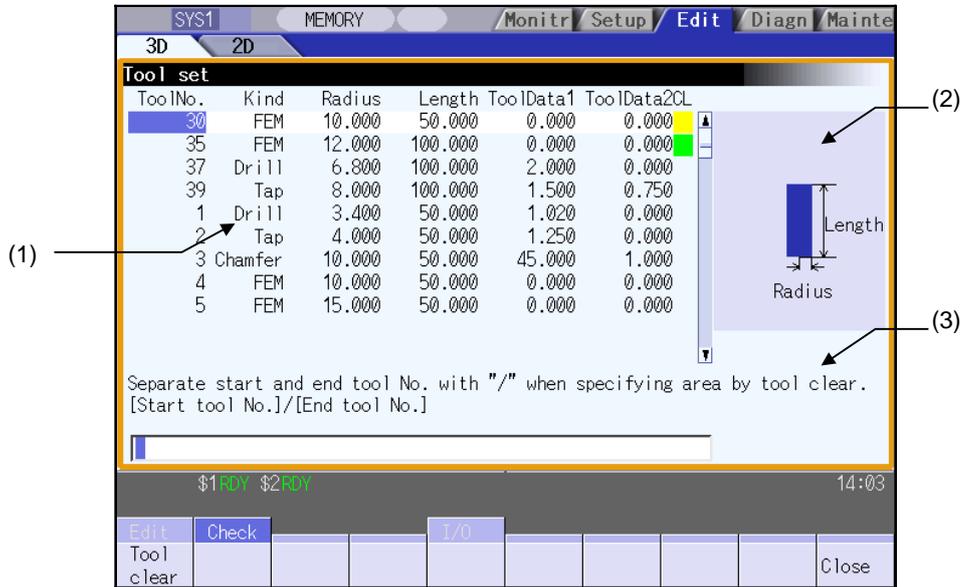
(Example)

[Parameter] #8920 = 1 [Tool compensation] No.1 : Length 150.000 L wear 0.000 Radius 20.000 R wear 0.000 [Workpiece coordinate] G59 : X 50.000 Y 50.000 Z 0.000 G54.1P1 : X 200.000 Y 200.000 Z 0.000 [Workpiece shape setting] Workrefpoint (I,J,K) = (0.000, 0.000, -200.000) WorkWidth (I,J,K) = (500.000, 500.000, 100.000)	[Machining program] N1 G28 XYZ N2 T1 M6 N3 G43 H1 N10 G0 G90 G59 X100. Y100. Z0. N11 G91 G01 Z-200. F100 N12 G02 I-15 N13 G01 Z200. N20 G0 G90 G54.1 P1 X200. Y200. Z0. N21 G91 G01 Z-200. F100 N22 G02 I-15 N23 G01 Z200. M30 %
--	---

CoordinateSystem	Coord Select	Drawing result when 3D check in example	
Machine Coord	All wk. coord	 <p>Basic machine coordinate system zero point</p>	<p>Arranges the workpiece based on the machine coordinate zero point. Draws the tool movement of all workpiece coordinate systems on the basic machine coordinate system.</p>
Work Coord	All wk. coord	 <p>Workpiece coordinate zero point G59</p> <p>Workpiece coordinate zero point G54.1P1</p>	<p>Arranges the workpiece based on the workpiece coordinate zero point. Draws the tool movement of all workpiece coordinate systems, considering workpiece coordinate zero points, G59 and G54.1P1, to be at the same position.</p>
Work Coord	G59	 <p>Workpiece coordinate zero point G59</p>	<p>Arranges the workpiece based on the selected workpiece coordinate zero point. Only draws the tool movement of the workpiece coordinate G59.G54.1P1 is not drawn.</p>

4.4.10 Setting the Tool Shape

Press the menu key to display a pop-up window similar to the following, and set the tool shape used in the solid display.



Display items

Display item	Details
(1) Tool shape setting area	This sets all tool shape items.
(2) Guide drawing	This displays a tool shape guide drawing.
(3) Tool clear guidance area	This displays a guidance concerning the area of tool clear.

Menus (when the cursor is at the tool number)

Menu	Details	Type	Reference
<input type="text" value="Tool clear"/>	This clears the tool shape data where the cursor is located. Specification method: Tool No. to start/end tool clear (Example) 10/30: Data in the lines of tool No.10 to 30 will be cleared. 10/E: Data in the lines of tool No.10 and after will be cleared. When the <input type="text" value="INPUT"/> key is pressed without specifying tool No., the data in the line at the current cursor position will be cleared.	A	"Clearing the tool number"
<input type="text" value="Close"/>	This closes the pop-up window and quits this function.	C	

Menus (when the cursor is at Kind (tool type))

Menu	Details	Type	Reference
Default tool	This sets the tool type. A drill displays if the default tool is selected.	C	"Setting the tool type"
Ball endmill			
Flat endmill			
Drill			
Bulnosed endmill			
Chamfer			
Tap			
Close	This closes the pop-up window and quits this function.	C	

Menus (when the cursor is at Radius (tool radius), Length (tool length), or ToolData1,2 (shape data))

Menu	Details	Type	Reference
Close	This closes the pop-up window and quits this function.	C	

Menus (when the cursor is at CL (tool color))

Menu	Details	Type	Reference
Default color	This sets the tool color. This displays in red if the default color is selected.	C	"Setting the tool color"
Gray			
Red			
Yellow			
Blue			
Green			
Light blue			
Purple			
Pink			
Close	This closes the pop-up window and quits this function.	C	

Registering the tool data

A maximum of 80 tools can be registered.

- (1) Use the cursor keys to move the cursor to the tool number setting column.
The cursor can be moved freely from the 1st to the 80th line regardless of the number of tools being registered.

- (2) Input the Tool No.  "10" appears at the cursor position.
The cursor moves one place to the right.
(Ex.) 10

- (Note 1)** Register a new tool data to set the default value for the tool type, tool radius, tool length, and shape data 1,2, and tool color.
(Note 2) If two or more of the same tool number is registered, the tool shape data that is registered first is selected.
(Note 3) When an unregistered tool is selected, the previously used tool is used as is.

Clearing the tool data (Clearing tool data for one line at the cursor position)

<When inputting directly>

- (1) Use the cursor keys to move the cursor to the tool number setting column.
- (2) Input as shown below.  The contents of the line where the cursor is located are all cleared.
The cursor does not move.
0

<When using menu operations>

- (1) Use the cursor keys to move the cursor to the tool number setting column.
- (2) Press the menu .  The menu is highlighted.
- (3) Press the key.  Background color of the line at the cursor position is turned to light blue.
The operation message "Erase?(Y/N)" is displayed.
- (4) Press the or key.  The contents of the line where the cursor is located are all cleared.
Background color of the line at the cursor position is returned.
The cursor does not move.
The menu highlight is returned to normal.
Press any key other than and to cancel.

Clearing the tool data (Clearing tool data for multiple lines)

- (1) Press the menu . The menu is highlighted.
- (2) Set the first and last tool No. of the area to be cleared by dividing by "/" and press the key. . The background color of the lines for tool No. 20 to 50 is highlighted in light blue. The operation message "Erase? (Y/N)" appears. When the tool data to be cleared does not exist within the page currently being displayed, the cursor is moved to the first line of the area to be cleared, making it the top of the display area. Specify the last line "E" to make it the end of the clearance range. (Example) 10/E
- (3) Press the or key. or Press any key other than and to cancel. The contents of the line highlighted in light blue are all cleared. Background color is returned. The menu highlight is returned to normal.

(Note 1) A setting error occurs when the specified tool No. does not exist.

(Note 2) A setting error occurs when the specified ending tool No. does not exist in the lower line than the starting tool No.

(Note 3) When the area to be cleared ranges over 2 pages, tools in the specified area in the second page and after will be cleared, as well.

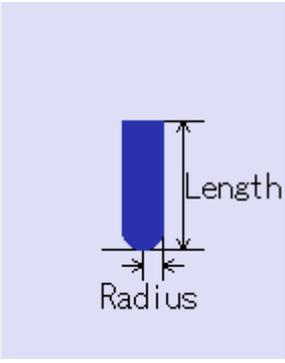
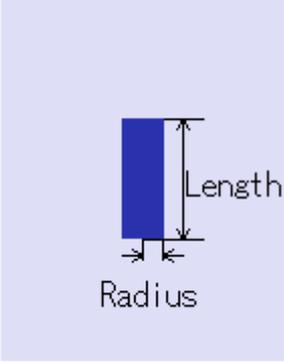
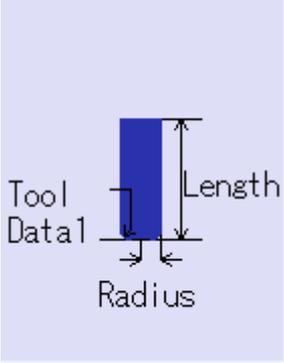
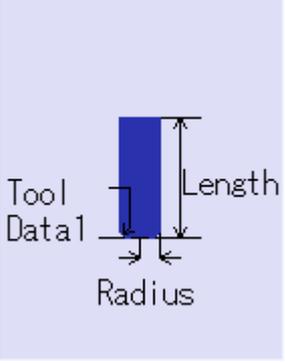
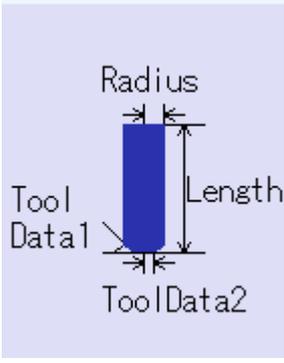
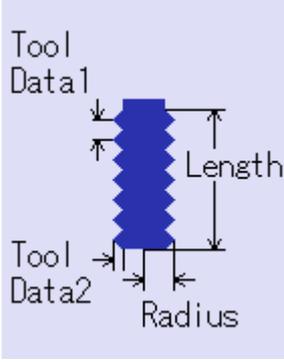
(Note 4) When two or more of the same tool Nos. are specified, tool data of the tool number registered first is cleared.

Setting the tool type (Kind)

- (1) Use the cursor keys to move the cursor to the tool type setting column.
- (2) Press the tool type menu. . "Ball endmill" appears at the cursor position. The cursor moves one place to the right. The ball end mill guide drawing (note) appears in the guide drawing area.
- It is also possible to set input values.
- 0 : Default tool
 - 1 : Ball end mill
 - 2 : Flat end mill
 - 3 : Drill
 - 4 : Bull nose end mill
 - 5 : Chamfer
 - 6 : Tap

(Note) The drill appears in the guide drawing area when the default tool is designated.

Tool type guide drawing

1: Ball end mill	2: Flat end mill	3: Drill
		
4: Bull nose end mill	5: Chamfer	6: Tap
		

Setting the tool color (CL)

Set the tool color for each tool.

- (1) Use the cursor keys to move the cursor to the tool color setting column.
- (2) Press the tool color menu.
(Ex.) Menu



The cell where the cursor is located appears in green. The cursor moves to the tool number column one line below.

It is also possible to set input values.

- 0 : Default color
- 1 : Gray
- 2 : Red
- 3 : Yellow
- 4 : Blue
- 5 : Green
- 6 : Light blue
- 7 : Purple
- 8 : Pink

(Note) The tool appears in red when the default color is designated.

4.4.11 Availability of Check Mode with Other Functions

Refer to "4.3.9 Availability of Check Mode with Other Functions".

4.4.12 Handling of Variable Commands, Parameter Input by Program, and Compensation Data Input by Program

Refer to "4.3.10 Handling of Variable Commands, Parameter Input by Program, and Compensation Data Input by Program".

4.4.13 Notes for Program Check (3D)**Check function**

- (1) The check function draws while referring only to the NC internal operation results. Thus, the path is not drawn for commands requiring machine movement.
- (2) Check search and check start are not possible during automatic start or automatic pause. The operation message "Executing automatic operation" appears.
- (3) Operation search is possible for a program with 32 or less file name characters.
- (4) Graphic check is not available for MDI program.

Notes for checking (Common for continuous check and check step)

- (1) The drawing is not initialized and the program check is not interrupted even if the menu , , or key is pressed while checking.
- (2) Check reset is applied when changing to another screen while checking.
- (3) Automatic operation cannot be performed even if the automatic operation button is pressed while checking. An operation alarm ("Program check mode") occurs.
Execute automatic start after completing the check or after stopping the check by pressing the .
- (4) Refer to "4.3.10 Handling of Variable Commands, Parameter Input by Program, and Compensation Data Input by Program" for further details on all types of data set in the program while checking.
- (5) The program stops if there is a M00/M01 command. However, when executing the M01 command, drawing is stopped regardless of the ON/OFF status of the optional stop switch signal.
- (6) The tool shape is drawn using the value set in "Tool set" window (Radius, Length, ToolData1,2).
As for the tool moving path, select with the parameter "#8920 3D tool Ofs Select", whether the tool compensation amount (radius dimension/length dimension) in Tool compensation screen is added to the path or not.
When the compensation amount is added, pay an attention so that the compensation No. during tool length/radius compensation and T command value is the same number. Failure to observe this could result in over cutting or insufficient cutting because the tool shape does not match the machine position.
- (7) If the following data for new tool is "0" or no setting when changing the tool, the new tool is drawn with the default setting.
 - Type : "Drill" is selected when "Kind" is not set.
 - Length : 150mm (5.90 inch) when "Length" is "0".
 - Radius : 20mm (0.79 inch) when "Radius" is "0".
 - Color : "Red" is selected when "CL" is not set.
- (8) When the parameter "8920 3D tool Ofs Select" is set to "0" or "1", tool is drawn with its radius the default value (20mm) if the tool radius to be displayed is "0" or less.

- (9) The menu related to graphic function cannot be selected during the automatic operation with the trace mode ON.
- (10) The zero point mark is not drawn.
- (11) The contents of ONB display and buffer display in the check screen during automatic operation are the contents of the program during automatic operation. The contents of ONB display and buffer display in the operation screen during check execution are the contents of the program during check execution.
- (12) When operation search or check operation ("Check continu" or "Check step") is performed during program restarting, the operation message "in program restarting" is displayed, and the program will not be searched or graphically checked.

Notes for ending check

- (1) If an M02/M30 command is found, "Program check completed" occurs.
- (2) The check mode ends when check reset, the trace display is turned ON, etc.
Use these operations to clear the error when a program error occurs.
When the NC has been reset while the check screen is displayed, graphic check will be completed, but the menu for changing from 2D to 3D graphic check will be remained invalid. Press the Check reset menu to validate the switching operation.

Tool replacement method

- (1) Set the standby tool count in the parameters "#1327 3D ATC type". The tool replacement method with each parameter is as follows. (Tool replacement commands other than M6 (M22, etc.) are not supported.)

#1327 3D ATC type		0	1	2
Machining program	Header section	T1;	T1; T2;	
	Machining process section	M6T2; Drawing by tool T1	M6T3; Drawing by tool T1	M6T1; Drawing by tool T1
		M6T3; Drawing by tool T2	M6TXX; Drawing by tool T2	M6T2; Drawing by tool T2
		M6TXX; <small>(Note 1)</small> Drawing by tool T3	M6; Drawing by tool T3	M6T3; Drawing by tool T3
End section	M6;	M6;	M6TXX;	

(Note 1) TXX is the T number for the empty pot.

Drawing position

(1) Set the drawing position with the parameter "#8920 3D tool Ofs Select".

With the parameter "#8920 3D tool Ofs Select", select which data is used for calculation of the drawing position (tool center point), the tool compensation amount stored in the NC memory or the data set in the "Work set" window. In the table below;

"NC memory data" indicates "the compensation amount stored in the NC memory".

"3Ddata" indicates "the data set in the "Tool set" window".

Setting of #8920	Tool shape data used for display		Tool center position (Z axis)
0 (Tool measurement type I)	Radius compensation	NC memory data	In G43 modal: Machine position - 3Ddata
	Length compensation	3D data	In G44 modal: Machine position - 3Ddata
1 (Tool measurement type II)	Radius compensation	NC memory data	In G43 modal: Machine position - NC memory data
	Length compensation		In G44 modal: Machine position + NC memory data (Note) This setting cannot be used with tool measurement type I.
2 (Tool measurement type I)	Radius compensation	3D data	In G43 modal: Machine position - 3Ddata
	Length compensation		In G44 modal: Machine position - 3Ddata
3 (Tool measurement type II)	Radius compensation	3D data	In G43 modal: Machine position - 3Ddata
	Length compensation		In G44 modal: Machine position - 3Ddata (Note) This setting cannot be used with tool measurement type I.

(Note 1) As for the 1st and 2nd axis (basic axis I,J) and the 3rd axis (basic axis K) during G49 (tool compensation cancel mode), the machine position is applied to the drawing position.

(Note 2) In the workpiece coordinate system drawing, "Machine position" in above table means "Workpiece coordinate position".

4.5 Program Input/Output



The machining program can input/output between the NC internal memory and the external input/output device. The hard disk inside the NC unit is also treated as an external device.

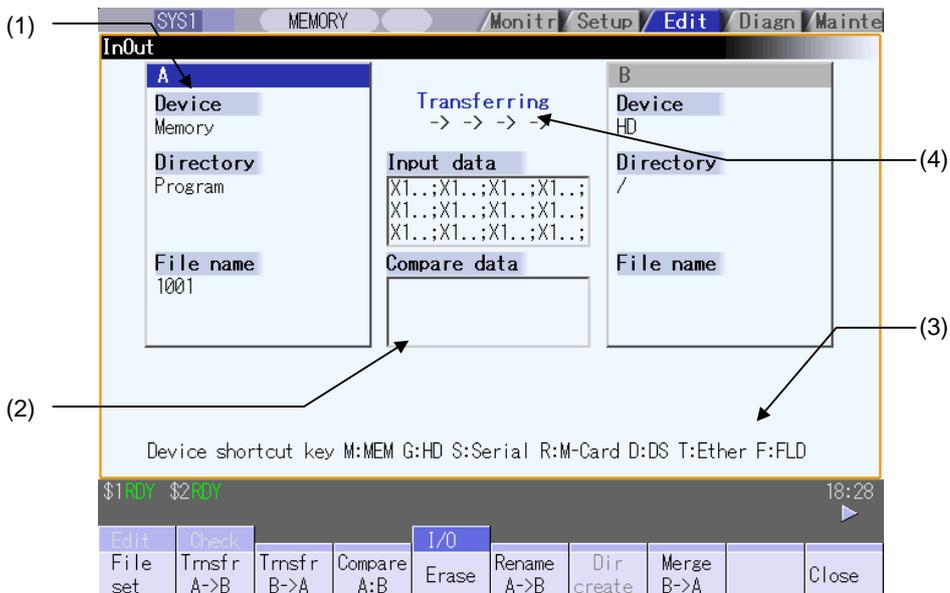
(Note) When the program input/output function is used, set the parameter "#8923 Hide Edit-IO menu" to "0".

This function applies to the following devices.

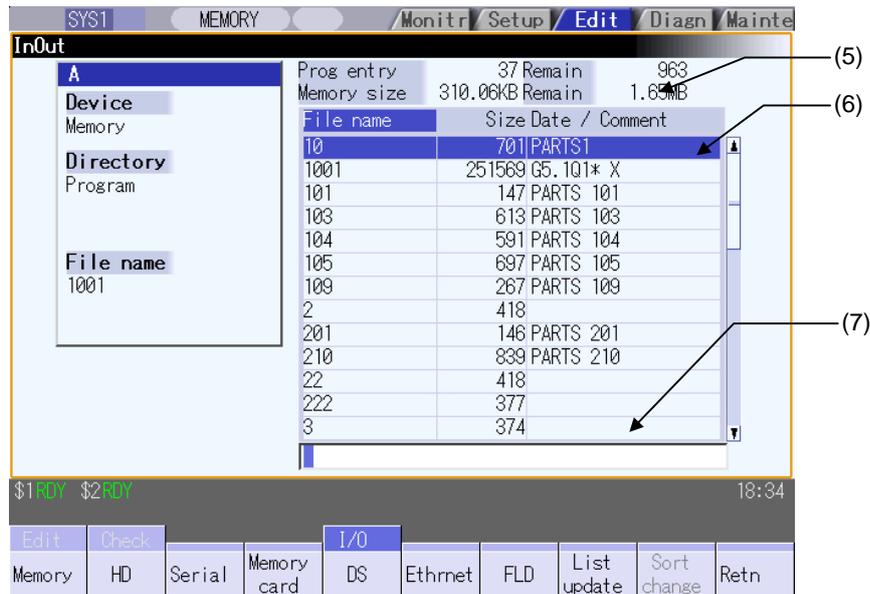
- (1) Memory (NC internal memory)
- (2) HD
- (3) Serial
- (4) Memory card (front IC card)
- (5) DS (compact flash at NC)
- (6) Ethernet
- (7) FD
- (8) Anshin-net server

(Note) In 70 series, only the memory, the serial, the memory card, Ethernet and Anshin-net server can be used.

<During file transmission>



<During file setting>



Display items

Display item	Details
(1) Device, directory, file name setting area	This sets the device, directory, and file name for which the transmission, compare, and erase functions are used. When the number of characters exceeds the display possible number (directory:63, file name:42), the excess is not displayed.
(2) Input/compare data display	This displays data that is being transferred or compared. If an error occurs while comparing data, the block for which the error occurred displays.
(3) Guidance display area	This displays the device name shortcut key.
(4) Process progress display section	This displays the details and data input/output direction for the process currently being performed.
(5) Capacity display section (Note 1)	This displays the file (machining program) registration count information and memory capacity information for the selected device. Number of programs registered : This displays the file (machining program) registration count. Remainder : This displays the remaining file registration count. This displays only when the memory is selected. Number of memory characters : This displays the memory character count. Remainder : This displays the remaining character count.
(6) List display section	This displays the device A or B directory contents list (directory and file name). File name : This displays the file (machining program) name. Size : This displays the file size. <DIR> displays in the case of the directory. Date/comment : The file comment (max. 17 characters) displays in the case of the memory device. The file update data displays in the case of the HD, memory card, DS, FD, or Ethernet devices.
(7) Input area	This displays the input key.

(Note 1) Some items may not be displayed depending on the device.

Display item \ Device	Memory	HD	Serial	Memory card	DS	Ethernet	FD	Anshin-net server
Number of programs registered	○	○	×	○	○	○	○	×
Remainder	○	×	×	×	×	×	×	×
Number of memory characters	○	○	×	○	○	○*	○	×
Remainder	○	○	×	○	○	×	○	×
List	○	○	×	○	○	○	○	×

○ : Displayed × : Not displayed

- When the Ethernet parameter "#97*1 Host n no total siz" is set to 1, the number of host memory characters will not appear.

(Note 2) When the device is not mounted, a registration count and a memory capacity are not displayed and an empty list is displayed.

(Note 3) When the device is serial, an empty list is displayed.

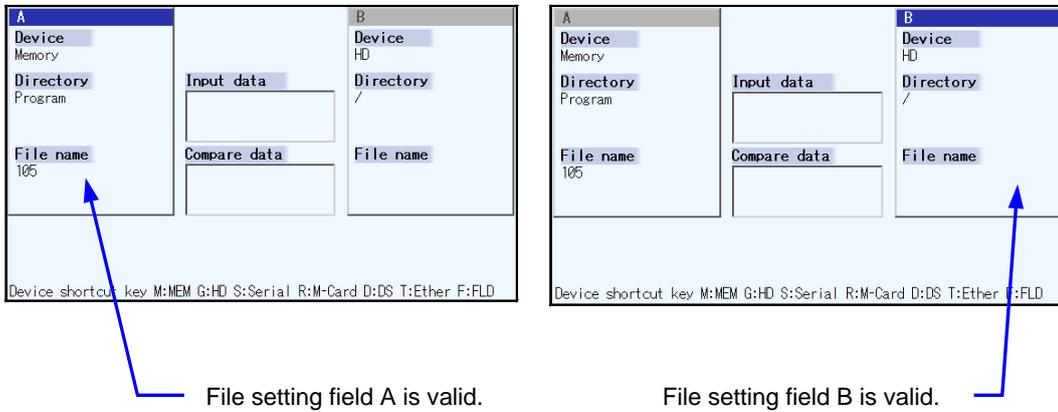
Menus

Menu	Details	Type	Reference
File set	This sets the device, directory and file name for which input/output operations are performed. (Note) When the device is serial or nothing, an empty list is displayed. When the device is not mounted, an operation message "(Device name) not ready" appears.	A	4.5.2 Selecting a Device, Directory, and File
Transfr A→B	This copies the file in file setting column A to file setting column B. (The originally transferred file is not affected.)	A	4.5.3 Transferring a File
Transfr B→A	This copies the file in file setting column B to file setting column A. (The originally transferred file is not affected.)	A	
Compare A:B	This compares the files in file setting column A and file setting column B.	B	4.5.4 Comparing Files (Compare)
Erase	This erases the selected file in file setting column.	A	4.5.5 Erasing a File
Rename A→B	This changes the name of the file in file setting column A to that of in file setting column B. (Note) The same device must be selected for A and B.	A	4.5.6 Changing a File Name (Rename)
Dir create	This creates a new directory in the selected file setting column. The directory can be created when HD, memory card, DS or FD is selected for the device.	A	4.5.7 Creating a Directory
Merge B→A	This adds the file contents of file setting column B to file setting column A. (The file setting column B file is not affected.)	A	4.5.8 Merging a File
Close	This closes the pop-up window and quits this function.	C	
MemCrd format	The formats the memory card (front IC card).	A	4.5.9 Formatting an External Device
DS format	This formats the DS. This menu is only for 700 series.	A	
FD format	This formats the FD. This menu is only for 700 series.	A	
Warning cancel	This cancels a warning from network service.	C	4.5.14 Sharing Machining Data "Canceling a warning from network service"
Stop	This interrupts the process (transfer, compare, etc.) during its execution.	C	

4.5.1 Changing the Valid Area

It is necessary to enable the areas including device, directory, and file name in the file setting column A or B in order to set those items.

Change the display area using the , direction cursor keys or the , tab keys.



Changing file setting column B to a valid area (refer to upper right diagram)

- (1) or key. File setting column B changes to a valid area.

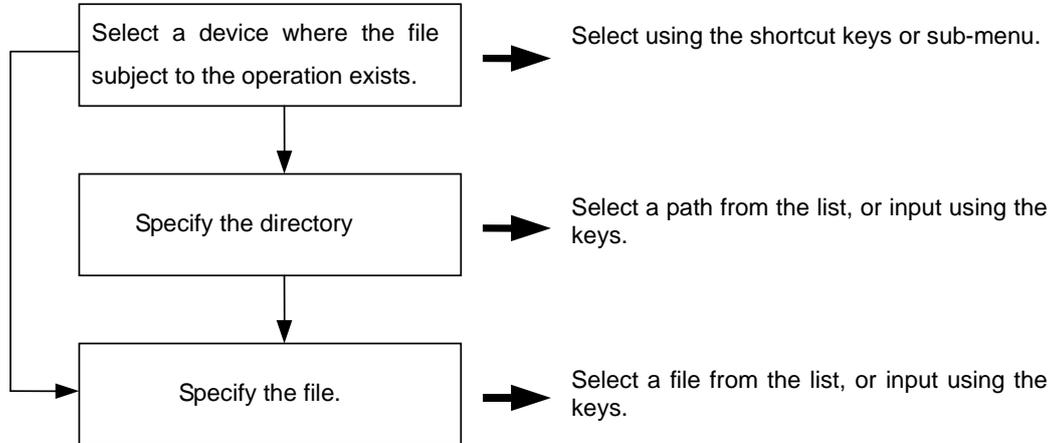
Changing file setting column A to a valid area (refer to upper left diagram)

- (1) or key. File setting column A changes to a valid area.

4.5.2 Selecting a Device, Directory, and File

This section explains the file device, directory, and file name specification methods for the file transfer and erase commands etc. performed on this screen.

File operation procedure



Outline of device, directory, and file name designating methods

Device	Designation target file	Designation method		
		Device	Directory	File name
NC memory	<ul style="list-style-type: none"> • Machining program • User macro program • Fixed cycle program 	Select from the submenu	- (Default)	Key input in the input area, and then <input type="text" value="INPUT"/>
		Designate using the shortcut key.		Select from the list
Device other than the NC memory	All files	Select from the submenu	Key input in the input area, then <input type="text" value="INPUT"/>	Key input in the input area, and then <input type="text" value="INPUT"/>
		Designate using the shortcut key.	Select from the list	Select from the list

- (1) The device can be selected using the sub-menu or shortcut keys.
(The devices that can be used will differ depending on the specifications.)
- (2) One of the following methods can be used to designate the directory (for devices other than the NC memory) and file name.
 - Designate the directory path or file name in the input area and press the key.
 - Move the cursor in the list display section to the target directory or file name and press the key.
 A wildcard (*) can be used for the file name.

(Note) When the number of characters of directory path (full path) exceeds 128, changing directories cannot be performed.

Notes when selecting a file

- (1) During directory and file name setting, the designated directory, path or file name will be set, even if it does not actually exist. This will not cause an error. Note that the previously set directory is overwritten.
- (2) When selecting a fixed cycle program, it is necessary to set the basic common parameter "#1166 fixpro". Furthermore, select "Memory" for the device, and "Program" for the directory.

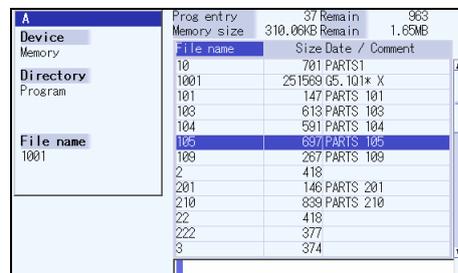
Initial directory when the Ethernet is selected

An initial directory is specified at turning the power ON by the following parameter.

- #9706 Host No.
- #9714 Host1 directory
- #9734 Host2 directory
- #9754 Host3 directory
- #9774 Host4 directory

Operation method (Selecting an NC memory program)

- (1) Press the **M** short cut key. → The device name changes to "Memory".
"Program" appears in the directory.
- (2) Press the menu **File set**. → The NC memory program list and input area appear.



The menu appears as follows.

<First page>

Memory	HD	Serial	Memory card	DS	Ethernet	FD	List update	Sort change	Retn
--------	----	--------	-------------	----	----------	----	-------------	-------------	------

<Second page>

Comment nondisp		Program	Program all			Anet server			Retn
-----------------	--	---------	-------------	--	--	-------------	--	--	------

(When specifications of all devices is valid.)

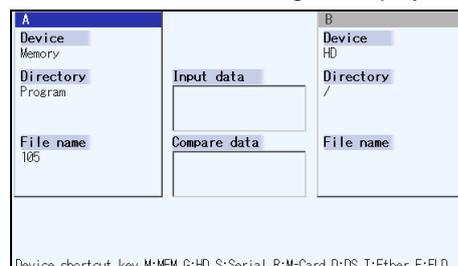
By pressing the operation menu at this point, it is possible to change the device.

When the menu **Program** or **Program all** is selected, the setting of "File name" column is cleared.

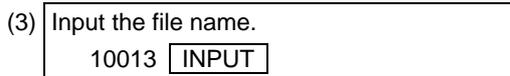
When the device is only the memory, the menu **Program** or **Program all** is valid.

When selecting the file name from the list

- (3) Move the cursor to the file name to be selected, and fix.
↑, ↓, INPUT → The selected file name appears.
The list and input area are cleared, and the screen returns to the original display.
The menu returns to its original display.



When inputting the file name from the input area:



The input file name appears.
The list and input area are cleared, and the screen returns to the original display.
The menu returns to its original display.

Designating multiple files

A wildcard (*) can be used for the file name.

By using a wildcard, multiple files can be transferred, compared, and erased at one time.

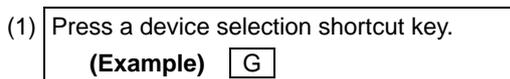
[Ex. 1] When programs 1 to 1000 exist in the NC memory.

- Designate "*" in the file name : The target program is "1 to 1000".
- Designate "*.*" in the file name : There is no target program.
- Designate "1*" in the file name : The target program is "1, 10 to 19, 100 to 199, 1000".
- Designate "1*.*" in the file name: There is no target program.
- Designate "**1" in the file name : The target program is that where the position of the 1 is "1".
(1, 11, 21, 31,101, 111,981, 991)
- Designate "**1*" in the file name : The target program is all of the programs containing "1".
(1, 10 to 19, 21, 31,100 to 199, 201, 210,981, 991,1000)
- Designate "1*1" in the file name: The target program is "11, 111, 121, 131, 141, 151, 161, 171, 181, 191".

[Ex. 2] When programs 1.PRPG to 1000.PRPG, and 1 to 1000 exist in the HD.

- Designate "*" in the file name : The target program is "1 to 1000".
- Designate "*.*" in the file name : The target program is "1.PRPG to 1000.PRPG".
- Designate "1*" in the file name : The target program is "1, 10 to 19, 100 to 199, 1000".
- Designate "1*.*" in the file name: The target program is "1.PRPG, 10.PRPG to 19.PRPG, 100.PRPG to 199.PRPG, 1000.PRPG".

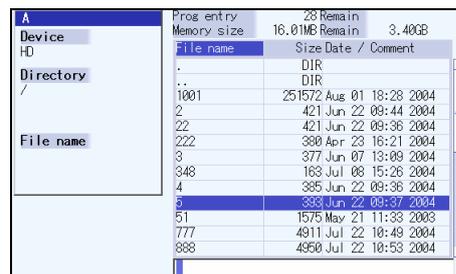
Operation method (Selecting a file from a device other than the NC memory)



The device name changes to "HD".
The root directory (/) appears in the directory.



The HD program list and input area appear.



A menu similar to the following appears.

<First page>

Memory	HD	Serial	Memory card	DS	Ethernet	FD	List update	Sort change	Retn
--------	----	--------	-------------	----	----------	----	-------------	-------------	------

<Second page>

Comment nondisp		Program	Program all			Anet server			Retn
-----------------	--	---------	-------------	--	--	-------------	--	--	------

(When specifications of all devices is valid.)

By pressing the operation menu at this time, the device can be changed.

When the device is only the memory, the menu , are valid.

When selecting a directory from the list.

- (3) Move the cursor to the directory to be selected and fix.
, ,

Repeat this operation until the cursor arrives at the target directory.
 Next, refer to operation (4).



The directory selected in the directory column appears.
 The contents of the selected directory appear in the list.
 Select ".." to move to one directory above.
 There is no change when "." is selected.

When inputting a directory from the input area:

- (3) Input the directory path.
(Example) /PRG/PRECU



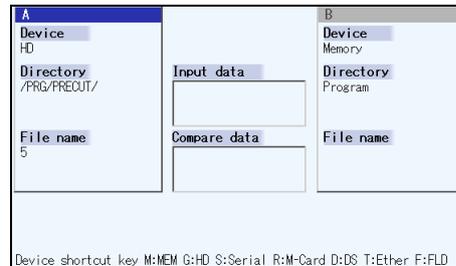
The input directory path appears in the directory column.

When selecting the file name from the list.

- (4) Move the cursor to the file name to be selected and fix.
, ,



The file name selected in the file name column appears.
 The list and input area are cleared, and the screen returns to the original display.
 The menu display returns to normal.



When inputting a file name from the input area:

- (4) Input the file name.
 10013.PRG



The file name input in the file name column appears.
 The list and input area are cleared, and the screen returns to the original display.
 The menu display returns to normal.

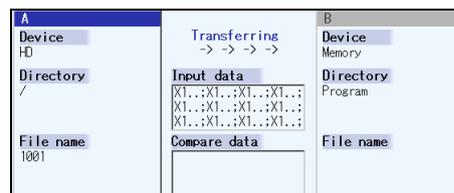
4.5.3 Transferring a File

This section explains the method used to transfer files between different devices, or between the same device.

Refer to "4.5.2 Selecting a Device, Directory, and File" for details of the device, directory, and file specification method.

Operation method

- (1) Use the or key to select file setting column A.
- (2) Designate the transfer source device, directory and file name. ➔ The designated file displays in file setting column A. A wildcard "*" can be used for the file name to designate multiple files.
- (3) Use the or key to select file setting column B.
- (4) Designate the transfer destination device, directory and file name. ➔ The designated file appears in file setting column B.
- (5) Press the menu . ➔ A message appears to confirm the transfer.
Press the menu when transferring from device B to device A.
- (6) Press the or key. ➔ File transfer is commenced. The process progress display area displays message indicating that transfer is in progress, and also indicates the transfer direction. Furthermore, the data being transferred appears in the input/comparison data display area of the input data display column.



A message appears indicating a transfer completion after transfer has been completed.

CAUTION



";", "EOB", "%", and "EOR" are symbols used for explanation. The actual codes for ISO are "CR, LF" ("LF") and "%".

The programs created on the Edit screen are stored in the NC memory in a "CR, LF" format, however, the programs created with external devices such as the FD or RS-232C may be stored in an "LF" format.

The actual codes for EIA are "EOB (End of Block)" and "EOR (End of Record)".



To prevent the influence of data loss and data transformation over the line, always carry out data comparison after transferring a machining program.

Notes (Notes related to transferring in general)

- (1) Depending on the type of file, some data cannot be transferred during automatic operation. Do not transfer during automatic operation.
- (2) When the capacity of the transfer destination is reached during file transfer, only the data transferred up to that point is registered as a file, and an error will occur.
- (3) During input to the NC memory or comparison, if the file format size on the NC memory side differs from the other side file format size (when the maximum number of registrations differs between the NC memory and the other side), processing is carried out matching the smaller size.
(Ex. 1) If a format size of 200 files is input for a format size of 1000 NC files, 200 files are registered.
(Ex. 2) If a format size of 1000 files is input for a format size of 200 NC files, the files up to the 200th file are registered and an error message appears. (The remaining files are not registered.)
- (4) Up to 223 files, including the directory, can be registered in the FD's root directory.

Notes (Notes when transferring machining program files)

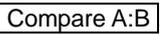
- (1) For the serial, always set both ends of the "EOR" code at the head and end to feed (NULL). If "EOB" etc., is directly after "EOR", the operation may not be executed normally due to the input buffer influence during the next input operation.
- (2) The transfer speed is slower if there are many registrations.
- (3) The size of one block of the machining program to be transferred should be 250 characters or less.
- (4) When using tape, carry out parity V adjustment to improve the reliability of the tape format. Then use it with the input/output parameter "Parity V" validated.
- (5) When the machine tool builder macro and fixed cycle program are input, change the program type with the parameter (#1166 fixpro). Also, set the device and directory as follows.
Device: Memory, Directory: Program
- (6) Transferring or verifying the multiple files between the external device connected serially and that other than the serial connection.
- (7) With machining program created by the MELDAS500 Series or earlier model, "EOB" is registered as "LF". However, when these programs are stored in the 700/70 Series NC memory, "EOB" will be converted to "CR LF", and the number of characters will increase. Thus, when all of the machining programs output from an MELDAS500 Series or earlier model, having the same specifications as the maximum memory capacity, are stored in the 700/70 Series NC memory, the memory capacity may be exceeded.
- (8) When the file to be transferred (input) is running or in "program restarting" mode, the operation message "Executing automatic operation" or "Program restarting" is displayed and file will not be transferred (input).

4.5.4 Comparing Files (Compare)

This chapter explains method used compare files after transferring.

Refer to "4.5.2 Selecting a Device, Directory, and File" for details of the device, directory, and file specification method.

Operation method

- (1) Use the  or  key to select file setting column A.
- (2) Designate the device, directory and file name to be compared.  The designated file name appears in file setting column A.
- (3) Use the  or  key to select file setting column B.
- (4) Designate the other device, directory and file name to be compared.  The designated file name appears in file setting column B.
- (5) Press the menu .  The file comparison starts.
The input data and comparison data appear in the input/comparison data display area.
A message appears when the comparison is completed.
If a comparison error occurs, the block with the error is displayed in the comparison data display column on the screen.

Input data
.;X1.;X1.;X1.;X1.;X1
.;X1.;X1.;X1.;X1.;X1
.;X1.;X1.;X1.;X1.;X1
Compare data
.;X1.;X1.;X1.;X1.;X1
.;X1.;X1.;X1.;X1.;X1
.;X1.;X1.;X1.;X1.;X1

(Note) Files that can be compared are text files only.
Correct outcome will not be obtained through binary file comparison.

4.5.5 Erasing a File

This chapter explains the method used to delete files.

Refer to "4.5.2 Selecting a Device, Directory, and File" for details of the device, directory, and file specification method.

Operation method (Deleting a file in file setting column A)

- (1) Use the or key to select file setting column A.
- (2) Designate the device, directory and file name to be erased.  The designated file appears in file setting column A.
- (3) Press the menu .  A message to confirm the erasing appears.
- (4) Press the or key.  The file is erased.
A message appears when the erasing is completed.

(Note) If the file to be erased is running, an error will occur, and the file will not be erased.

Operation method (Deleting a file in file setting column B)

Use the or key to select file setting column B, and designate the device, directory, and file name for file setting column B. Following this, the operation method is same as for "Deleting a file in file setting column A".

4.5.6 Changing a File Name (Rename)

This section explains the method used to change the file name.

Refer to "4.5.2 Selecting a Device, Directory, and File" for details of the device, directory, and file specification method.

(1) Changing a file name

- (1) Use the or key to select file setting column A.
- (2) Designate the device, directory and file name before the change.  The designated file appears in file setting column A.
- (3) Use the or key to select file setting column B.
- (4) Designate the device, directory, and file name after the change.  The designated file appears in file setting column B.
- (5) Press the menu.
- (6) Press the or key.  Renaming of the file starts.
A message appears when the renaming is completed.

(Note 1) Make sure the original and new devices are the same.

(Note 2) If the file to be renamed is running or program restarting, an error will occur, and the name will not be changed.

(Note 3) If a file that does not exist is designated for the original file, or if an existing file name is designated for the new file selection, an error occurs, and the name is not changed.

(Note 4) The directory where the file in operation search/check search is stored cannot be renamed. The operation message "Can't rename designated file" appears.

4.5.7 Creating a Directory

This section explains the method used to newly create a directory.

Refer to "4.5.2 Selecting a Device, Directory, and File" for details of the device, directory, and file specification method.

Operation method

(1) Use the or key to select file setting column A.

(2) Designate the device



The designated device name appears.
The devices that can create a directory are HD, FD, Memory card and DS.

(3) Select a directory to be newly created from the list, or designate by inputting from the input area.



The designated details appear.

Device	Memory size	6 Remain	3.40GB
HD	1.54GB	Remain	
Directory	File name	Size	Date / Comment
/PRG/PREOUT/	.	DIR	
	..	DIR	
	2	421 Jun 22 08:44 2004	
	3	377 Jun 07 13:09 2004	
	4	385 Jun 22 08:36 2004	
	5	393 Jun 22 08:37 2004	

(4) Set the new directory in the input area and press key.



The set directory name appears.

(5) Press the menu.



The directory is created.

The directory can also be created in file setting area B.

(Note1) Up to 223 files, including the directory, can be registered in the FD's root directory.

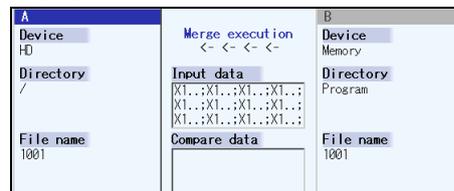
(Note2) Make sure that the directory path is less than 100 characters. When the path exceeds 100 characters, it cannot be recognized as a path.

4.5.8 Merging a File

This section explains the method used to add a file in file setting column B to a file in file setting column A. Refer to "4.5.2 Selecting a Device, Directory, and File" for details of the device, directory, and file specification method.

Operation method

- (1) Use the or key to select file setting column A.
- (2) Designate the merge destination device, directory, and file name. → The designated file name appears in file setting column A.
- (3) Use the or key to select file setting column B.
- (4) Designate the merge source device, directory, and file name. → The designated file name appears in file setting column B.
- (5) Press the menu . → A message confirming the merge appears.
- (6) Press the or key. → The file merge starts.
The data being merged appears in the input/comparison data display area input data display column.
Furthermore, a message indicating that merging is being performed, and an arrow from the merge source (file setting column B) to the merge destination (file setting column A) display in the process progress display area.



A message appears when the merge is completed. The details of the file in file setting column A will be as follows when the merge is completed.

(FILE A) G28 XYZ G90 F800 G00 X100. Y100.;	}	Details of file in file setting column A before merge.
(FILE B) G91 G28 XYZ F1000; G01 X200. Y200.;		
M02; %	}	Details of file in file setting column B before merge.

The details of the file in file setting column B do not change.

(Note 1) If the merge destination file (file in file setting column A) is running, an error will occur, and the files will not be merged.

(Note 2) The devices that can be merged are the memory, HD, FD, memory card and DS.

4.5.9 Formatting an External Device

This section explains the method to format the external devices.

Operation method (Formatting an FD) [700 series only]

- (1) Insert a floppy disk in the FD drive, and press the menu → A message confirming the formatting appears.
- (2) Press the or key. → The FD is formatted.
A message appears when the formatting is completed.

(Note 1) The FD is formatted with FAT (1.44MB).

(Note 2) The volume label is set when formatting the FD.

Operation method (Formatting a memory card)

- (1) Press the menu . → A message confirming the formatting appears.
- (2) Press the or key. → The memory card is formatted.
A message appears when the formatting is completed.

(Note 1) The memory card is formatted with FAT16.

(Note 2) The volume label is set when the memory card is formatted.

Operation method (Formatting a DS) [700 series only]

First, press the menu . The following operations are the same as "Formatting a memory card".

(Note 1) The DS is formatted with FAT16.

(Note 2) Only the DS formatted with FAT16 can be used. The DS with NTFS cannot be used.

(Note 3) As for the DS formatted with NTFS, reformat it with FAT formatted by Windows to use.
(NC cannot convert NTFS partition to FAT formatted.)

(Note 4) The volume label is not set even when the DS is formatted.

4.5.10 List of File Names

There is a directory for each type of data in the NC memory.

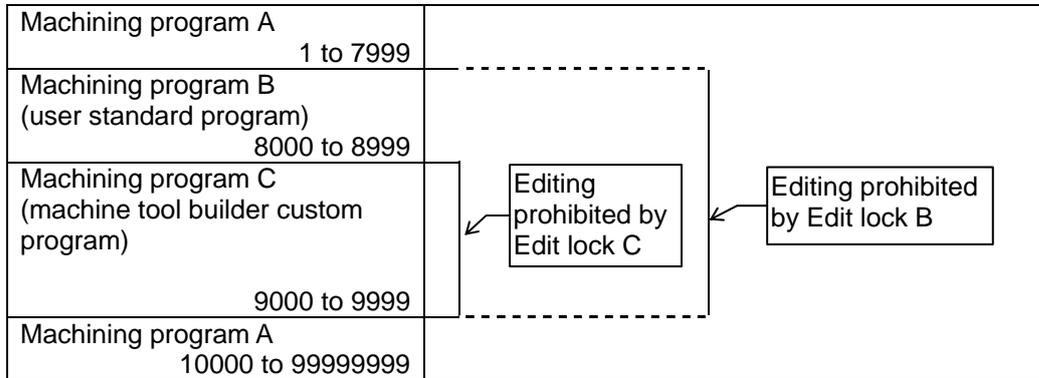
The NC memory directory and file name handled on this screen is as follows.

Furthermore, when storing the NC memory file in a device other than the NC memory, be careful not to change the file extension (.XXX).

Data type	NC memory directory path	File name
Machining program	/PRG/USER	(Program No.)
Fixed cycle program	/PRG/FIX	(Program No.)

4.5.11 Edit Lock B and C

This function prohibits editing, erasing, etc. of the machining programs B and C, and protects the machining programs stored in the NC memory.



The operations below in the Edit MDI and the Input/Output screens are influenced by the edit lock setting. An error will occur if operations that are not possible are attempted. When the edit lock is valid, processing is executed (except the edit lock target program) by the input/output function.

○ : Operation possible × : Operation not possible

Screen	Operation	Edit lock B			Edit lock C		
		Machining program			Machining program		
		A	B	C	A	B	C
Edit	Search	○	○	×	○	○	×
	Edit	○	×	×	○	○	×
	MDI registration	○	×	×	○	○	×
Edit (Input/Output)	Transfer	○	×	×	○	○	×
	Compare	○	×	×	○	○	×
	Copy	○	×	×	○	○	×
	Merge	○	×	×	○	○	×
	Rename	○	×	×	○	○	×
	Erase	○	×	×	○	○	×
Operation	Buffer correction	○	×	×	○	○	×

(Note) When the multiple part system specification is valid, the machining programs of all the part systems are protected by edit lock B and C.

4.5.12 Program Display Lock C

This function prohibits display and search for machining program C (machine tool builder custom program). The following influences are received on each screen according to parameter "#1122 pglk_c".

Influences of program display lock parameter on operation screen

#1122 pglk_c Screen operation	1 (Display:○ Search:×)	2 (Display:× Search:×)
Buffer display of operating program	The program contents are not displayed. Only "%" is displayed.	
Display when Program display highlights on 2D check full-screen		
When the menu Edit was pressed	The operation message "Program display lock C" appears. The window is opened.	
When the main menu P corr was pressed	(When the edit lock B or C is valid, edit lock B or C is given to priority.)	
Search from the operation search window	The search can be executed.	The search cannot be executed.
Search from the restart search window		The operation message "Program display lock C" appears. The window is not closed. The contents of input area are not cleared.
Search from the top search window (of restart search)		

Influences of program display lock parameter on operation screen

#1122 pglk_c Screen operation	1 (Display:○ Search:×)	2 (Display:× Search:×)
The file is automatically opened when changing the edit screen (Note 1)	The program contents are not displayed. Only "%" is displayed.	- (It is not possible to search, so the file is never automatically opened when changing the edit screen.)
When the file is opened by the menu Open .	The operation message "Program display lock C" appears. The window is not closed. (When the edit lock B or C is valid, edit lock B or C is given to priority.)	
When the main menu P corr was pressed	The program contents are not displayed. Only "%" is displayed.	
Right side program display when check main menu is selected		
Search from the operation search window of 2D/3D check	The search can be executed.	The search cannot be executed. The operation message "Program display lock C" appears. The window is not closed.

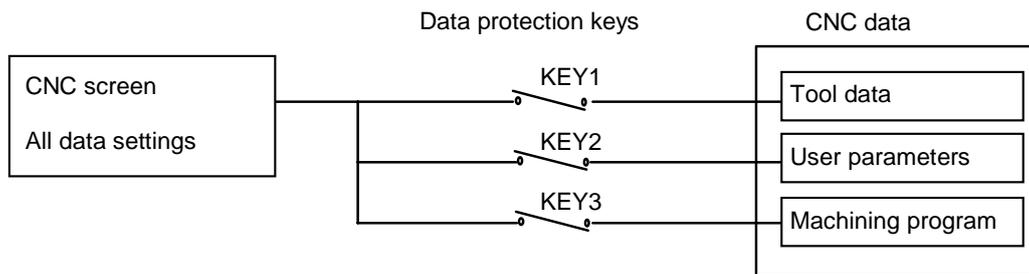
(Note 1) When changing the edit screen, the file opened immediately before or operation searched/check searched is automatically opened.

4.5.13 Data Protect Keys

The data protect keys can be used to prohibit data setting and erasure. The following three keys are available. (Their names differ according to the machine tool builder. For further details, refer to the instruction manual issued by the machine tool builder.)

- (1) KEY1: Provides general protection for tool data and protection of the coordinate system preset using the origin set.
- (2) KEY2: Protects user parameters and common variables.
- (3) KEY3: Protects the machining program.

Data settings and deletion is stopped if any of the data protection keys is OFF.



Machining program protection (KEY3)

When the KEY3 is OFF, the following operations are stopped.

No.	Operation details	Screen
1	MDI data memory registration	Edit
2	Machining program editing	Edit
3	New machining program creation	Edit
4	Comment setting for programs being registered	Edit
5	Machining program registration to memory, verification, input/output	Edit
6	Machining program deletion (1 program, all)	Edit
7	Comment setting for programs being registered	Edit
8	Machining program copy, file name change, merge	Edit
9	Machining program buffer correction	Operation

(Note 1) If editing or settings, etc. are performed on the screens shown above when KEY3 is OFF, a "Data protect" message appears.

(Note 2) When the multiple part system specification is valid, editing and input/output of the machining programs of all the part systems are prohibited by KEY3.

4.5.14 Sharing Machining Data

Machining data can be send to/received from Anshin-net server in the call center.

Operation method

- | | |
|--|---|
| (1) Use the <input type="text" value="←"/> or <input type="text" value="←"/> key to select file setting column A. | |
| (2) Designate the transfer source device, directory and file name. | ➔ The designated file appears. |
| (3) Use the <input type="text" value="→"/> or <input type="text" value="→"/> key to select file setting column B. | |
| (4) Press the menu <input type="text" value="File set"/> . | ➔ The designated file appears in file setting column B. |
| (5) Press the menu <input type="text" value="Anshin net"/> to designate the transfer destination device and file name. | ➔ "Anshin-net server" is displayed as device name.
The file name, unless specified, is as same as the name of the originally transferred file.
Only numbers within 10 characters can be used for a file name. |
| (6) Press the menu <input type="text" value="Transfr A→B"/> . | ➔ A message appears to confirm the transmission of machining data. |
| (7) Press the <input type="text" value="Y"/> or <input type="text" value="INPUT"/> key. | ➔ Transmission of machining data starts.
A message appears upon the completion of transmission. |

(Note 1) Neither HD operation nor memory card operation can be performed during file transmission from HD.

(Note 2) Only one file can be saved on Anshin-net server. The file previously exists on the Anshin-net server is overwritten.

(Note 3) Anshin-net icon appears in the upper middle of the screen during Anshin-net communications.

(Note 4) When Anshin-net server is specified as device name, the directory button is displayed in gray and cannot be selected.

Receiving machining data from Anshin-net server

- (1) Use the or key to select file setting column A.
- (2) Press the menu .
- (3) Press the menu to designate the transfer source device and file name.  "Anshin-net server" as device name and the designated file name are displayed.
- (4) Use the or key to select file setting column B.
- (5) Designate the transfer destination device, directory and file name.  The designated file appears.
- (6) Press the menu .  A message appears to confirm the reception of machining data.
When the file name is not designated in the file setting column A, the message "File name not designated for dev A" appears to indicate the error.
- (7) Press the or key.  The reception of machining data starts.
A message appears upon the completion of reception.

(Note 1) When the same file name exists in the transfer destination device, an error occurs and the transfer cannot be executed.

(Note 2) Anshin-net icon appears in the upper middle of the screen during Anshin-net communications.

(Note 3) When Anshin-net server is specified as device name, the directory button is displayed in gray and cannot be selected.

Erasing machining data on Anshin-net server

- (1) Use the or key to select file setting column A.
- (2) Press the menu .
- (3) Press the menu to designate the file name to erase. → "Anshin-net server", as device name, and the designated file name are displayed.
- (4) Press the menu → A message appears to confirm the erasing.
- (5) Press the or key. → The file will be erased.
A message appears upon the completion of erasing.

Canceling a warning from network service

- (1) Press the menu key → The warning message from network service is cleared.

(Note 1) A warning message from network service appears when any error occurs during communications.

(Note 2) NC reset can also clear the warning message from network service.

Notes

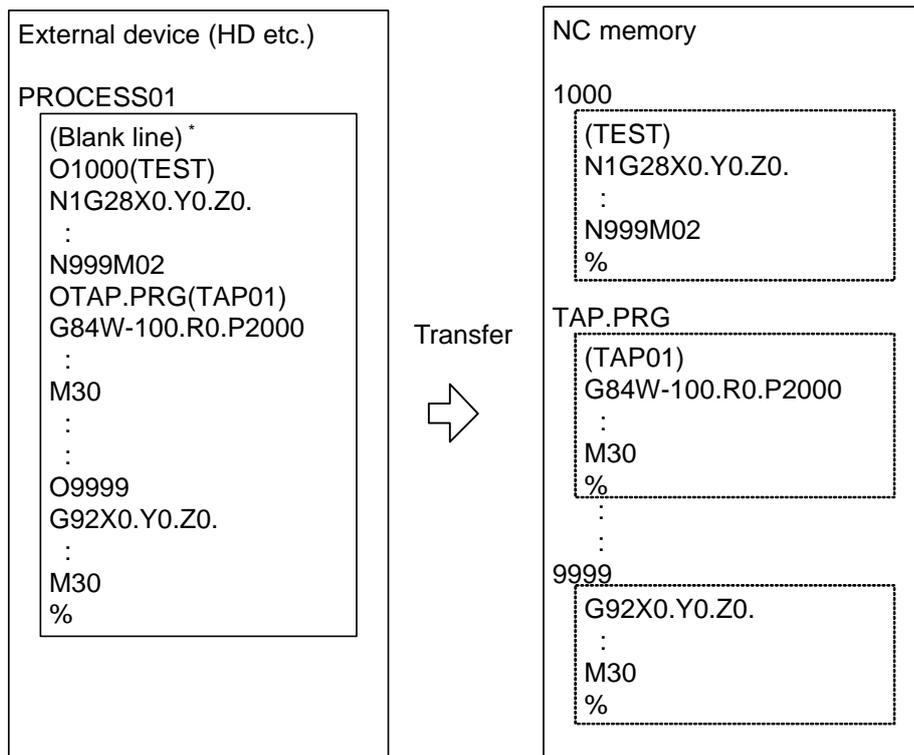
- (1) Transfer to/from Anshin-net server cannot be interrupted.
- (2) Only machining program, except fixed cycle and machine tool builder macro, can be transferred to Anshin-net server.
- (3) Data transfer between Anshin-net server and serial (RS232C) or Ethernet is not available.
- (4) The edit screen and the maintenance screen can share machining data. However, the machining data being transferred to/from either of the screens cannot be shared with the other screen.
- (5) Wildcard (*) cannot be used for the file name.
- (6) The capacity of machining program for transmission is restricted according to the settings of Anshin-net server.

4.5.15 The Batch Input/Output the Machining Program of NC Memory

One file which consists of two or more machining program can be transferred to NC memory by dividing the file. The machining programs united with one file can be transferred to the external device. The targets are the machining programs of the user.

Batch input and verify to NC memory

Example of "the external device -> NC memory" of machining program batch transfer



When the external device is serial, the top line is %.

When the transfer direction file name is designated, the top program name of transfer source can be omitted. ("O1000" can be omitted in above example.)

- (1) Use the or key to select file setting column A.
- (2) Designate the transfer source device and directory. The designated device and directory appear.
- (3) Designate the file which consists of two or more machining programs. The designated file name appears.
- (4) Use the or key to select file setting column B.
- (5) Designate the transfer direction device (memory). The designated device name (memory) appears.

- (6) Press the menu Program all.

➔

The "Program all" appears in the directory.

When the file name is designated, only the top program of transfer source is stored in the designated file.

For example of "Example of "the external device -> NC memory" of machining program batch transfer", when "TESTCUT" is designated as the transfer direction file name, the top program is transferred as not "1000" but "TESTCUT".
- (7) Press the menu Transfr A→B after press the menu Return or ◀ key.

➔

The message "OK? (Y/N)" appears.
- (8) Press the Y or INPUT key.

➔

Transmission of machining data starts. Transmitting data is displayed in input data display column. The message "Transfer complete" appears upon the completion of transmission.

When the same machining program already exists in transfer direction device, the message "Overwrite this file?(Y/N)" appears. The transferring the following program is continued by selecting Y or N key.
- (9) Press the menu Compare A:B key.

➔

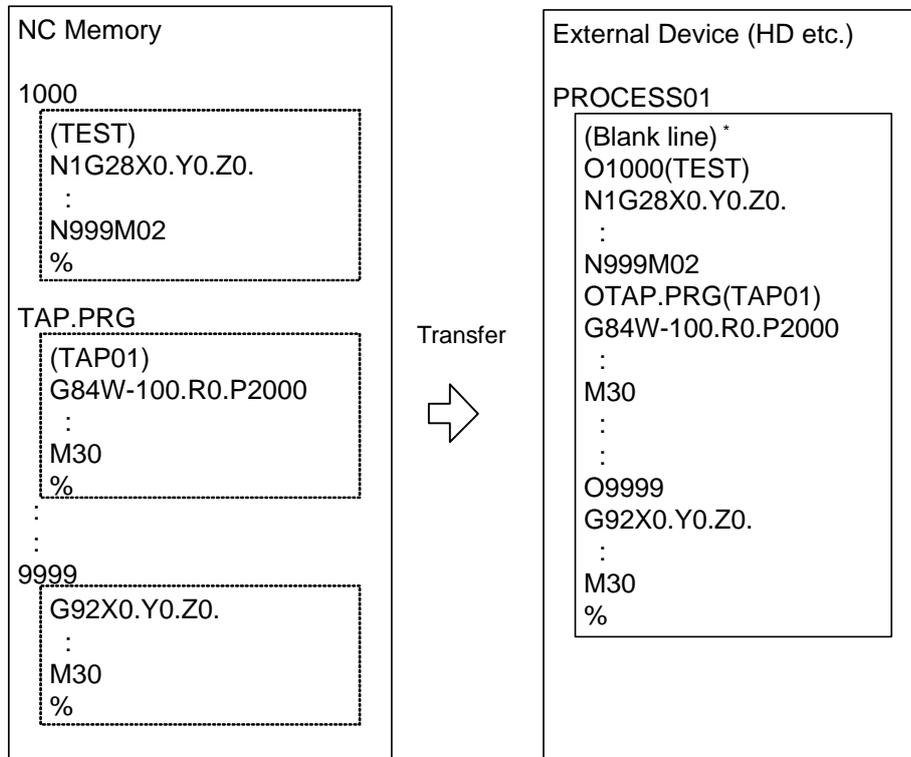
The verification of the file starts. The verifying data is displayed. When the verification is completed, the message appears. When the verification error occurs, the error block is displayed in verification data display column and the message "Compare error. Compare next file?(Y/N)" appears.

- (Note 1)** When programs are input in a batch to NC memory, always specify "Program batch" to NC memory directory as transfer direction. If "Program batch" is not specified, one file is transferred (No batch).
- (Note 2)** When the program which is subject to edit lock exists in transfer source, the message "Edit lock B" or "Edit lock C" appears, and transfer is interrupted. When the program which is not subject to edit lock is transferred, delete the program which is subject to edit lock in transfer source. (Refer to "4.5.11 Edit Lock B and C".)
- (Note 3)** When the machining program protection is valid (KEY3 is OFF), the program cannot be transferred/verified. (Refer to "4.5.13 Data Protect Keys".)
- (Note 4)** The program during automatic operation, program restart or program checking cannot be overwritten. the message "Executing automatic operation", "Program restarting" or "Program checking" appears, and transfer is interrupted.
- (Note 5)** The first line of the transfer source file is ignored.
- (Note 6)** When the transfer source is serial, the file is similarly transferred even if the transfer direction setting is the following case 1 and case 2.

	Case 1	Case 2
Device name	Memory	Memory
Directory	Program batch	Program
File name	(Null or file designation)	(Null or file designation)

Batch output and verify from NC memory

Example of "NC memory -> the external device" of machining program batch transfer



When the external device is serial, the top line is %.

- (1) Use the or key to select file setting column A.
- (2) Designate the transfer source device (memory). → The designated device (memory) appears.
- (3) Press the menu Program all. → The "Program all" appears in the directory.
Do not specify the file name. If the name is specified, an error will occur.
- (4) Use the or key to select file setting column B.
- (5) Designate the transfer direction device and directory. → The designated device name and directory appear.
- (6) Designate the transfer direction file to input multiple machining programs. → The designated file name appears.
When the file name is not designated, output to the file "ALL.PRG" of the designated directory.

- (7) Press the menu . → The message "OK? (Y/N)" appears.
- (8) Press the or key. → Transmission of machining data starts. Transmitting data is displayed in input data display column. The message "Transfer complete" appears upon the completion of transmission.

When the same machining program already exists in transfer direction device, the message "Overwrite this file?(Y/N)" appears. The transferring the following program is continued by selecting or key.
- (9) Press the menu key. → The verification of the file starts. The verifying data is displayed. When the verification is completed, the message appears. When the verification error occurs, the error block is displayed in verification data display column and the message "Compare error. Compare next file?(Y/N)" appears.

- (Note 1)** When programs are output in a batch from NC memory, always specify "Program batch" to NC memory directory as transfer source. If "Program batch" is not specified, the program is transferred by one program per one file.
- (Note 2)** When the programs which are subject to edit lock exists in NC memory, those file is not transferred. (The files are transferred excluding those files.) The verification is executed excluding the programs which are subject to edit lock. (Refer to "4.5.11 Edit Lock B and C".)
- (Note 3)** When the machining program protection is valid (KEY3 is OFF), the program cannot be transferred/verified. (Refer to "4.5.13 Data Protect Keys".)
- (Note 4)** When the transfer direction is serial, the program is similarly transferred even if source of following case 1 and case 2 is set.

	Case 1	Case 2
Device name	Memory	Memory
Directory	Program batch	Program
File name	(Null)	* (Wild card)

Notes

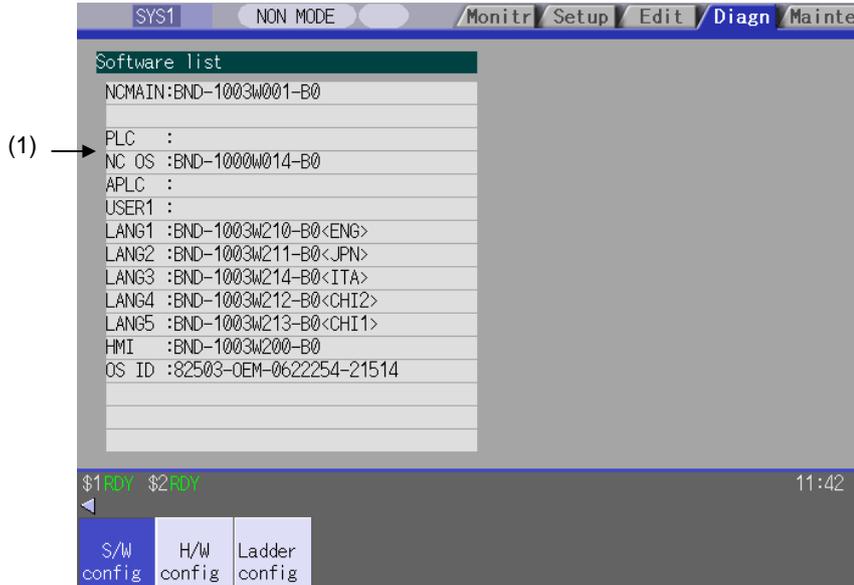
- (1) When the machining program batch input/output function of NC memory is used, "(" cannot be used for the machining program name. It is recognized that the inside of "(" is a comment.
- (2) The file name of file which consists of two or more machining program can have up to 32 characters, including the extension as well as the machining program.
- (3) When the "Program batch" is designated to NC memory directory, it is not possible to delete, rename and merge files. When the files are deleted in a batch, use "Program" for directory, and a wild card (*) for file name.
- (4) Refer to Note of "4.5.3 Transferring a File" for other notes.

5. Diagnosis Screens

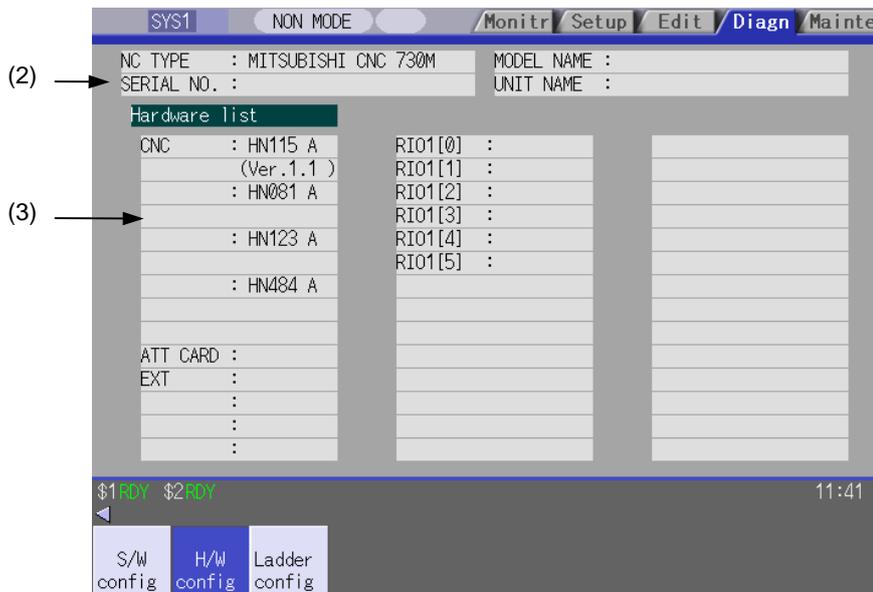
5.1 System Configuration Screen

The hardware configuration (card name and sub-number), software configuration (software number and sub-number), and PLC program configuration (file name, file label, and execution type) are displayed in this screen.

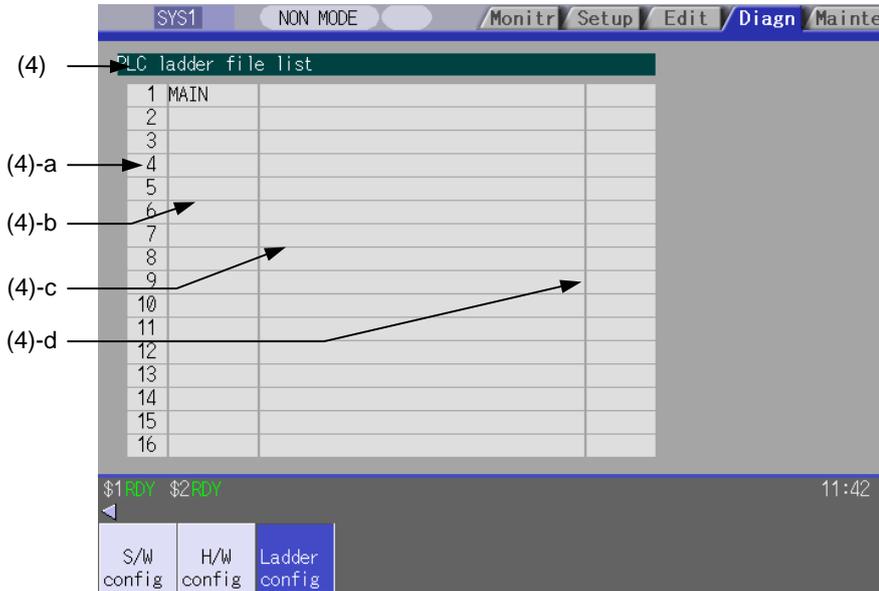
■ Software configuration



■ Hardware configuration



■ PLC program configuration



Display items

Display item	Details
(1) Software list	This displays a list of the software being used.
(2) NC serial No.	This displays the NC model name, serial No, model type, and unit type.
	NC TYPE: MITSUBISHI CNC 7*** NC type
	MODEL NAME: FCA730 Model name
	SERIAL NO.: M7123456789 Serial No.
	UNIT NAME: FCU7-MU011 Unit type
(3) Hardware list	This displays each hardware name.
70 Series	CNC : HN761 CPU card : HN451 Memory card
	EXT : HR751 PLC card (for type A) (Blank for type B)
	RI01[n] : Remote IO unit 1 (n= 0 to 7) RI02[n] : Remote IO unit 2 (n= 0 to 7) RI03[n] : Remote IO unit 3 (n= 0 to 7)
	There are up to three channels. The 7th and 8th station of the RIO3 channel is fixed for use with the handle I/F and is not displayed.

Display item	Details															
(3) Hardware list																
700 Series	CNC (Note) : HN115A : HN081A : HN116/122 : HN482/484	Main card with LANCPU Power card CPU card (Differs between M720, M730, M750) Memory card (Differs between M720, M730, M750) The CNC unit is composed of four PCBs. Bus connections are used with all cards.														
	ATT CARD :	Currently unused.														
	EXT : EX891 : HR553 : HR577	Back panel Extension unit Extension unit The extension unit is a hardware option. The PLC high-speed engine or PROFIBUS card, etc. is connected. The back panel + up to three cards are displayed.														
	RIO1[n] : RIO2[n] : RIO3[n] :	Remote IO unit 1 (n= 0 to 7) Remote IO unit 2 (n= 0 to 7) Remote IO unit 3 (n= 0 to 7) There are up to three channels. The 7th and 8th station of the RIO3 channel is fixed for use with the handle I/F and is not displayed.														
(4) PLC ladder file list	This displays the file name, file label, execution type for each PLC ladder program. Use  ,  keys to change the pages and refer it.															
(a) Registration No.	This displays the registration No. of each PLC program file. Registration size is max. 32 files. The target file to be executed is max. 20 files.															
(b) File name	This displays the file name of PLC program file. (data of GX Developer) Max. 8 characters.															
(c) File label	This displays this file label of PLC program file. (data of GX Developer) Max. 32 characters.															
(d) Execution type	This displays the execution type of PLC program. <table border="1" data-bbox="699 1294 1353 1496"> <thead> <tr> <th>Display</th> <th>Meaning</th> </tr> </thead> <tbody> <tr> <td>HIGH</td> <td>High-speed PLC program</td> </tr> <tr> <td>MIDDLE</td> <td>Middle-speed PLC program</td> </tr> <tr> <td>INITIAL</td> <td>Initial state of PLC program</td> </tr> <tr> <td>WAIT</td> <td>Standby PLC program</td> </tr> <tr> <td>LOW</td> <td>Low-speed PLC program</td> </tr> <tr> <td>(blank)</td> <td>Not the target of the execution.</td> </tr> </tbody> </table>		Display	Meaning	HIGH	High-speed PLC program	MIDDLE	Middle-speed PLC program	INITIAL	Initial state of PLC program	WAIT	Standby PLC program	LOW	Low-speed PLC program	(blank)	Not the target of the execution.
Display	Meaning															
HIGH	High-speed PLC program															
MIDDLE	Middle-speed PLC program															
INITIAL	Initial state of PLC program															
WAIT	Standby PLC program															
LOW	Low-speed PLC program															
(blank)	Not the target of the execution.															

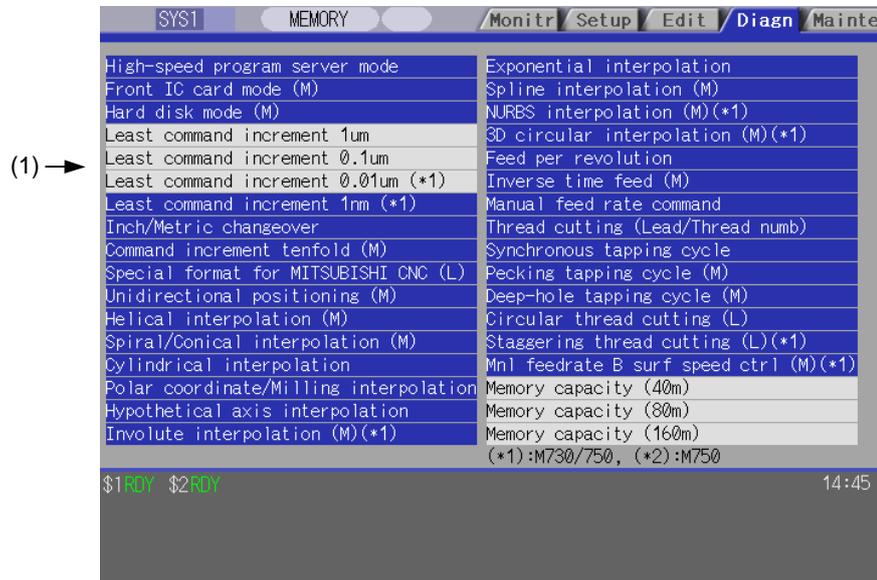
Menus

Menu	Details	Type	Reference
	This displays the software list.	C	
	This displays the hardware list.	C	
	This displays the PLC program list (file name, file label, and execution type)	C	

5.2 Option Display Screen

The contents of the options registered in the NC are displayed in this screen.

The option items are displayed by name. If all of the options cannot be seen in one screen, the rest of options can be displayed by pressing the page changeover key.



Display items

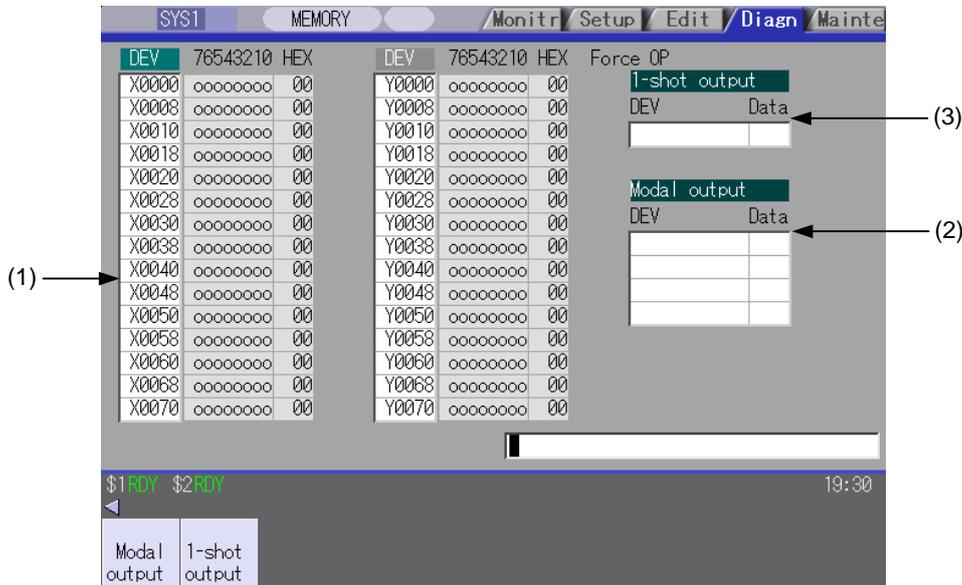
Display item	Details
(1) Option items	The list of currently usable options are displayed. As for the currently usable options, the background color is displayed in blue. The option set when the power supply was turned ON can be currently used.

5.3 I/F Diagnosis Screen

The various input/output signals for the PLC (Programmable Logic Controller) control can be displayed and set in this screen.

These signals can be used in confirmation of the machine sequence operation during PLC development, and in confirmation and forced output, etc., of the input/output data between the NC and PLC.

(Note) Pay close attention to the sequence operation when using these functions during machine operation.



Display items

Display item	Details
(1) Device No. and input/ output signal value (binary/ hexadecimal display)	This displays the data from the device Nos. designated in the setting area in numerical order. The data is displayed as binary (bit units) and hexadecimal values. Individual device Nos. can be displayed separately in the left area and right area. Select the valid area with the and \leftarrow key when \rightarrow operations such as display changeover and data setting are carried out. Each X, Y, M, F, L, SM, TI, TO, TS, TA, STI, STO, STS, STA, CI, CO, CS, CA, D, R, SB, B, V, SW, SD, W, P, K, and H data is the target data.
(2) Modal output	This displays the data and device to carry out modal output. The details to be defined are set here when carrying out the modal type forced output of PLC interface signals. Refer to "5.3.2 Carrying Out Modal Output" for details.
(3) 1-shot output	This displays the data and device to carry out one-shot output. The details to be defined are set here when carrying out the one-shot type forced output of PLC interface signals. Refer to "5.3.3 Carrying Out One-shot Output" for details.

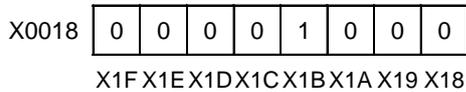
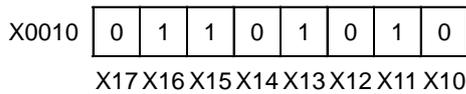
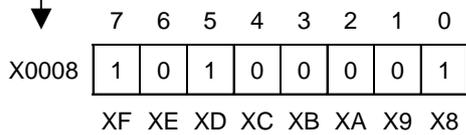
Menus

Menu	Details	Type	Reference
Modal output	This changes the setting area to an input standby status. The signal is forcibly output (modal).	A	5.3.2 Carrying out modal output
1-shot output	This changes the setting area to an input standby status. The signal is forcibly output (one-shot).	A	5.3.3 Carrying out one-shot output

How to read the device No. and display data

A device is an address for classifying a signal handled in the PLC. A device No. is a series of numbers attached to that device.

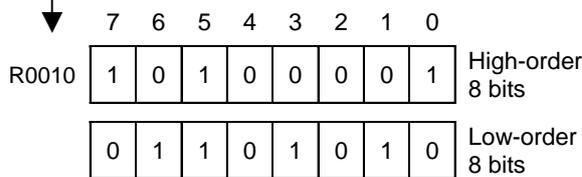
Device No. (bit-type)



DEV	76543210	HEX	DEV	76543210	HEX
X0000	00000000	00	Y0000	00000000	00
X0008	10100001	A1	Y0008	00000000	00
X0010	01101010	6A	Y0010	10100000	A0
X0018	00001000	08	Y0018	00000001	01
X0020	01111100	7C	Y0020	00000000	00
X0028	00001111	0F	Y0028	00000000	00
X0030	00000010	02	Y0030	00000000	00

Signal status display corresponding to each device No. (I/O)

Device No. (word-type)



DEV	76543210	HEX
R00010	10100001	A1
	01101010	6A
R00011	00001001	09
	00001001	05
R00012	00010010	12
	00001100	06
R00013	00001010	0A
	00010101	15

List of devices for PLC uses

Device	Device No.	No. of points	Units	Details
X (*)	X0 to X1FFF	8192	1-bit	Input signals to the PLC. Machine input, etc.
Y	Y0 to Y1FFF	8192	1-bit	Output signals to the PLC. Machine output, etc.
M	M0 to M10239	10240	1-bit	For temporary memory
F	F0 to F1023	1024	1-bit	For temporary memory. Alarm message interface.
L	L0 to L511	512	1-bit	Latch relay (Backup memory)
SM (*)	SM0 to SM127	128	1-bit	Special relay
TI	TI0 to TI703	704	1-bit	Timer contact
TO	TO0 to TO703	704	1-bit	Timer output
TS	TS0 to TS703	704	16-bit	Timer setting value
TA	TA0 to TA703	704	16-bit	Timer current value
STI	STI0 to STI63	64	1-bit	Integrated timer contact
STO	STO0 to STO63	64	1-bit	Integrated timer output
STS	STS0 to STS63	64	16-bit	Integrated timer setting value
STA	STA0 to STA63	64	16-bit	Integrated timer current value
CI	CI0 to CI255	256	1-bit	Counter contact
CO	CO0 to CO255	256	1-bit	Counter output
CS	CS0 to CS255	256	16-bit	Counter setting value
CA	CA0 to CA255	256	16-bit	Counter current value
D	D0 to D2047	2048	16-bit	Data register
R (*)	R0 to R13311	13312	16-bit	File register
SB	SB0 to SB1FF	512	1-bit	MELSEC NET/10 link special relay
B	B0 to B1FFF	8192	1-bit	MELSEC NET/10 link relay
V	V0 to V255	256	1-bit	MELSEC NET/10 edge relay
SW	SW0 to SW1FF	512	16-bit	MELSEC NET/10 link special register
SD	SD0 to SD127	128	16-bit	MELSEC NET/10 special register
W	W0 to W1FFF	8192	16-bit	MELSEC NET/10 link register

(Note) The use of devices marked with a * mark in the device column has been determined. Do not use devices other than those corresponding to the input/output signals with the machine side (input/output signals of the remote I/O unit), even if it is an undefined vacant device.

5.3.1 Displaying the PLC Device Data

The various status signals and register data used in the PLC can be monitored and displayed.

When this screen is first selected, the respective 16-byte amounts of input/output data are displayed from device "X0000" on the left display area, and from device "Y0000" on the right side.

This screen constantly monitors and displays the PLC signal statuses. Consequently, when signals are changed in the PLC, the display is changed according to the changes.

Note that because time differences occur between the PLC signal change and the signal display, there may be a delay in the display. The machine may also not be able to correspond to extremely brief signal changes.

Displaying the data of a arbitrary device No. "X0020"

(1) Press the menu tab key , , and select the area to display the data.

(2) Set the device No. (X0020), and press the  key.

➔ The device "X0020" appears at the head of the valid display area.

DEV	76543210	HEX	DEV	76543210	HEX
X0020	11111111	FF	Y0000	00000000	00
X0028	01000000	40	Y0008	00000001	01
X0030	00001110	0E	Y0010	00000000	00
X0038	00000000	00	Y0018	00000000	00

(Note) When setting the device No., an error will occur if a number exceeding the specifications or an illegal address is set.

Changing the display with the page keys

The valid area device Nos. change in page units when / is pressed. Changing of the pages stops within the range of device numbers of which the device has.

(1) Press the  key.

➔ The data is displayed from the next number currently displayed.

DEV	76543210	HEX	DEV	76543210	HEX
X0000	10000001	81	Y0000	00000000	00
X0008	00000010	02	Y0008	00000001	01
X0010	00001111	0F	Y0010	00000000	00
X0018	10011111	9F	Y0018	00000000	00



DEV	76543210	HEX	DEV	76543210	HEX
X0080	11111111	FF	Y0000	00000000	00
X0088	01000000	40	Y0008	00000001	01
X0090	00001111	0F	Y0010	00000000	00
X0098	00000000	00	Y0018	00000000	00

5.3.2 Carrying Out Modal Output

Modal type forced output of PLC interface signals is carried out. Once set, this data is held until cancelled, the power is turned ON/OFF, or other data is overwritten. There are four sets of devices that modally output. If this number is exceeded, the previously existing data is overwritten.

Menus used in modal output

Menu	Details	Type	Reference
Modal clear	This releases the modal output for the device at the cursor position in the modal output area. The released data is erased from this area.	C	"Releasing the modal output"

Modally outputting data "1" to device "X0048"

- (1) Press the menu key .  The modal output mode is entered, and the cursor appears at the modal output area.
- (2) Using the and keys, move the cursor to the setting position.
- (3) Set the device and data, and press the key.  Modal output is executed, and the cursor disappears. The data that was in the cursor position is overwritten by the input data, and is invalidated. The modal output mode is cancelled by pressing the key.
X0048/1

(Note 1) The data of the modally output device is displayed in order in the selected area.
This modal output is held until the output is cancelled or the power is turned OFF.

(Note 2) When no data is set (Ex. "X0048/", "X0048"), the operation message "Setting Data not found" is displayed.

Releasing the modal output

- (1) Press the menu key .  The modal output mode is entered, and the cursor appears at the modal output area.
- (2) Using the and keys, move the cursor to the data to be released.
- (3) Press the menu key .  The data that was in the cursor position is released from modal output. The "DEV" and "Data" columns become blank. The modal output mode is cancelled by pressing the key.

Caution

 Pay close attention to the sequence operation when carrying out forced data setting (forced output) in the I/F diagnosis screen during machine operation.

5.3.3 Carrying Out One-shot Output

The one-shot type PLC interface signal forced output is forcibly output only once during the screen operations. Thus, it may not be possible to confirm the PLC interface signals updated with the PLC on the screen.

One-shot outputting data "1" to device "X0042"

- | | | | |
|-----|--|---|---|
| (1) | Press the menu key 1-shot output . | ➔ | The one-shot output mode is entered, and the cursor appears at the one-shot output area. |
| (2) | Set the device and data, press the INPUT key.

X0042/1 INPUT | ➔ | The input data is overwritten in the one-shot output area, and is one-shot output. The cursor in the one-shot output area disappears. The data of the one-shot output device is displayed in order in the selected area. The one-shot output mode is cancelled by pressing the ◀ key. |

(Note 1) Because the input signal (X, etc.) to the PLC is updated at the head of each PLC cycle, the machine immediately returns to the normal state, even if one-shot type forced output is carried out.

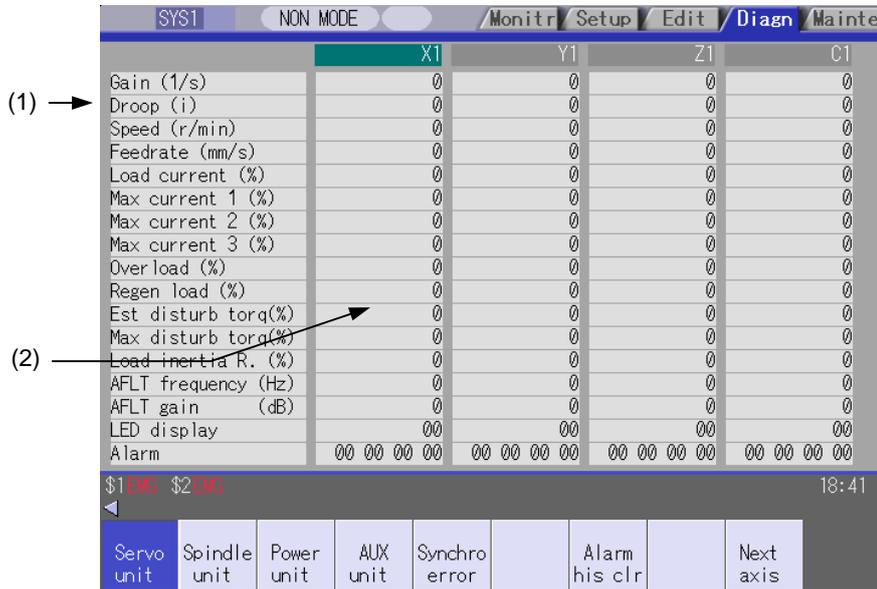
(Note 2) When no data is set (Ex.: "X0048/", "X0048"), the operation message "Setting Data not found" is displayed.

Caution

-  Pay close attention to the sequence operation when carrying out forced data setting (forced output) in the I/F diagnosis screen during mach

5.4 Drive Monitor Screen

The diagnosis information from the drive section can be monitored with this screen. Servo axis unit, spindle unit, power supply unit and synchronous error information is displayed.



Display items

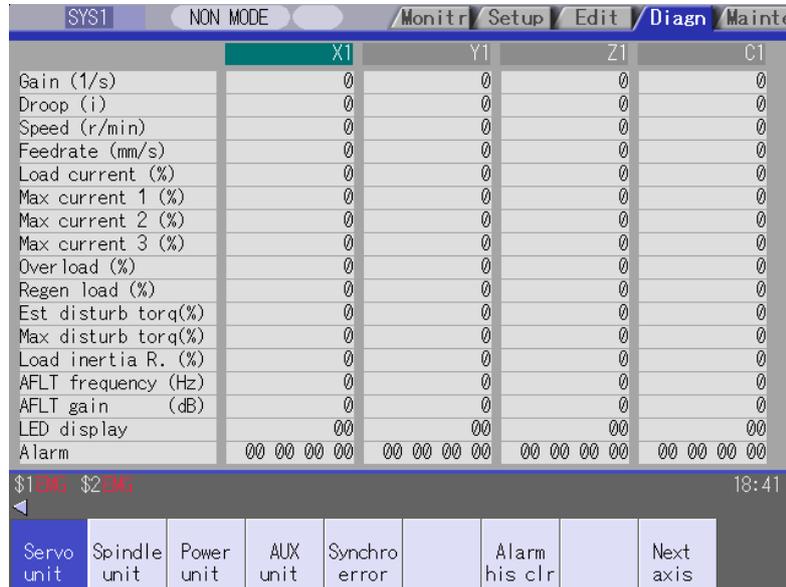
Display item	Details
(1) Monitoring items	This displays each item being monitored. The display is changed using the page changeover keys.
(2) Data of each axis and unit	This displays the data of each axis or each unit being monitored.

Menus

Menu	Details	Type	Reference
Servo unit	This displays the diagnosis information of the servo unit in the data display area.	B	5.4.1 Servo axis unit display items
Spindle unit	This displays the diagnosis information of the spindle unit in the data display area.	B	5.4.2 Spindle unit display items
Power unit	This displays the diagnosis information of the power supply unit in the data display area.	B	5.4.3 Display items for the power supply unit
AUX unit	This monitors the various data related to the auxiliary axis (MR-J2-CT) servo control. The menu appears and operation is possible only when there is one or more valid auxiliary axes in the auxiliary axis control. This menu is only for 700 series.	B	5.4.4 Display items for the auxiliary axis unit
Synchro error	This displays the diagnosis information of the synchronous error in the data display area. The menu appears and operation is possible only when the synchronous control axis option is valid.	B	5.4.5 Display items for the synchronous error
Alarm his clr	This clears the diagnosis information alarm history.	A	5.4.6 Clearing the alarm history
Next axis	This displays the data for the next four axes. The menu appears and operation is possible only when diagnosis information for five or more axes is displayed.	C	

5.4.1 Servo Axis Unit Display Items

The various data related to the servo axis (NC axis, PLC axis) is monitored. To reference, change the display items using the  key and  key.



The axis name set in the base axis specification parameter "#1022 axname2" appears at the axis name.

Display items

Display item	Details
Gain (1/s)	This displays the position loop gain. Position loop gain : $\frac{\text{Feedrate (mm/s)}}{\text{Tracking delay error (mm)}}$
Droop (i)	The error of the actual machine position to the command position is called droop. This error is proportional to command speed value.
Speed (r/min)	This displays the actual rotation speed of motor.
Feedrate (mm/s)	This displays the feedrate detected by the detector mounted on the machine end.
Load current (%)	This displays the FB value of the motor current in terms of continuous current during stalling.
Max current 1 (%)	This displays the motor current command in terms of continuous current during stalling. An absolute value of the current command peak value sampled after the power ON is displayed.
Max current 2 (%)	This displays the motor current command in terms of continuous current during stalling. An absolute value of the current command peak value sampled in most recent 2 seconds is displayed.
Max current 3 (%)	This displays the FB value of the motor current in terms of continuous current during stalling. An absolute value of the current FB peak value sampled in most recent 2 seconds is displayed.
Overload (%)	This is the data used to monitor the motor overload.
Regen Load (%)	This is the data used to monitor the resistance overload state when the resistance regenerative power supply is connected.
Est disturb torq (%)	This displays the estimated disturbance torque in terms of stall rated torque when the disturbance observer is valid.

Display item	Details
Max disturb torq (%)	This displays the estimated disturbance torque in terms of stall rated torque when the collision detection function is adjusted. An absolute value of the estimated disturbance torque peak value sampled in most recent 2 seconds is displayed.
Load inertia R. (%)	This displays the estimated load inertia ratio when the collision detection function is adjusted.
AFLT frequency (Hz)	This displays the present operation frequency of the adaptive filter.
AFLT gain (dB)	This displays the present filter depth of the adaptive filter.
LED display	This displays the 7-segment LED of the driver.
Alarm	This displays the alarms and warnings other than the LED display (displayed on drive unit side).
Cycle counter (p)	This displays the position within one rotation of the encoder detector. The position is displayed as a grid point value as "0", within one rotation in the range of "0" to "RNG (movement units) × 1000".
Grid space	This displays the grid space for the reference position return. (Command unit)
Grid amnt	This displays the distance from the dog-off point to the grid point when the dog-type reference position return is displayed. The grid mask amount is not included. (Command unit)
Machine posn	This displays the NC basic machine coordinate system position. (Command unit)
Motor end FB	This displays the feedback value of the speed detector. (Command unit)
Machine end FB	This displays the feedback position of the machine end position detector. (Command unit)
FB error (i)	This displays the error of the motor end FB and machine end FB.
DFB compen amnt (i)	This displays the compensation pulse amount during dual feedback control.
Remain command	The remaining movement distance of one block is displayed. (Command unit)
Currnt posn (2)	The value of the tool compensation amount subtracted from the current position is displayed. (Command unit)
Man int amt	The amount of interrupt movement in the manual absolute OFF state is displayed. (Command unit)
Abs pos command	This displays the coordinates of absolute position excluding the machine error compensation amount. (Command unit)
AUX current sta No. (Note)	This displays the current station No.
AUX current posn (Note)	This displays the current coordinates position.
AUX inst station No. (Note)	This displays the command station No. of automatic operation or the nearest station No. to stop by manual operation etc.
AUX inst posn (Note)	This displays the coordinates position corresponding to target station No.
Mach err comp val	This displays the machine error compensation amount. (Command unit)
Control input 1L 1H : 6L 6H	This indicates the control signal input from NC. This is used by the system.
Control output 1L 1H : 6L 6H	This indicates the control signal output to NC. This is used by the system.

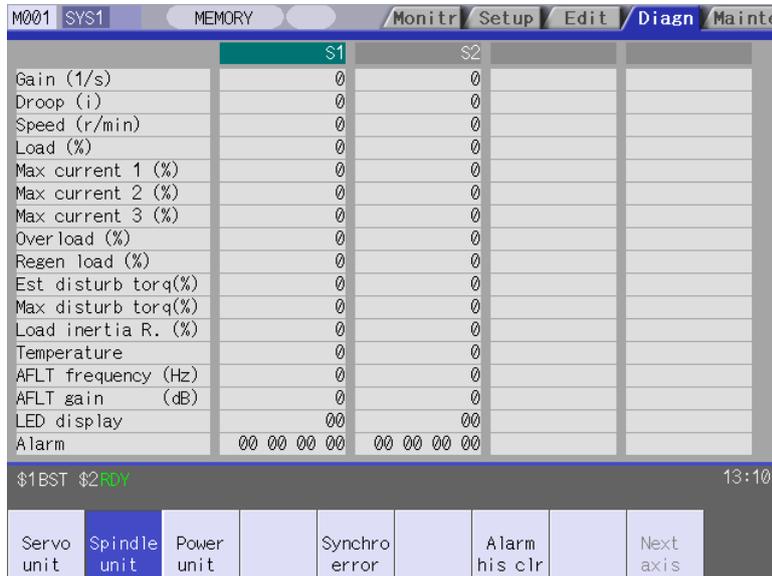
(Note) The data only for the PLC axis indexing function. "-" is displayed in the usual axis (NC axis or during PLC axis control) not used by the PLC axis indexing function.

Display item	Details
Detection system	This displays the detector type symbol of the absolute position detection system. ES : Semi-closed encoder EC : Ball screw end encoder LS : Linear scale MP : MP scale ESS : Semi-closed high-speed serial encoder ECS : Ball screw end high-speed serial encoder INC : Incremental
Power OFF posn	This displays the coordinate at NC power OFF in the basic machine coordinate system. (Command unit)
Power ON posn	This displays the coordinate at NC power ON in the basic machine coordinate system. (Command unit)
Current posn	This displays the current coordinate in the basic machine coordinate system. (Command unit)
R0	This displays the multi-rotation counter value of the detector stored in the memory during basic point setting.
P0	This displays the position within one rotation of the detector stored in the memory during basic point setting.
E0	This displays the absolute position error stored in the memory during basic point setting.
Rn	This displays the multi-rotation counter value of the current detector.
Pn	This displays the position within one rotation of the detector.
En	This displays the absolute position error during NC power OFF.
ABS0	This displays the absolute position reference counter.
ABSn	This displays the current absolute position.
MPOS	This displays the offset amount of the MP scale when the power is turned ON.
Unit type	This displays the servo driver type.
Unit serial No.	This displays the servo driver serial No.
Software version	This displays the servo side software version.
Control method	SEMI : Semi-closed loop CLOSED : Closed loop DUAL : Dual feedback
Motor end detector	This displays the motor end detector type.
Motor end detect No	This displays the motor end detector serial No.
Machine end detector	This displays the machine end detector type. The type is displayed when the control method is CLOSED or DUAL. * is displayed when the method is SEMI.
Mach. end detect No	This displays the machine end detector serial No.
Motor	This displays the motor type.
Work time	This displays the READY ON work time. (Units: 1hr)
Alarm hist 1: Time 1: Alarm : 8: Time 8: Alarm	This displays servo alarms that occurred in latest order with the following formats. Time : Work time when the alarm occurred. Alarm No. : Number of the servo alarms that occurred.
Maint hist 1 to 4	This displays the maintenance dates. Year : One digit Month : 1 to 9, X (Oct.), Y (Nov.), Z (Dec.)
Maint status	This displays the maintenance status.

5.4.2 Spindle Unit Display Items

The various data related to the spindle is monitored.

Change the display items using the  key and  key to refer to the data.



Display items

Display items	Details
Gain (1/s)	This displays the position loop gain. Position loop gain : $\frac{\text{Feedrate (mm/s)}}{\text{Tracking delay error (mm)}}$
Droop (i)	The error of the actual machine position to the command position is called droop. This error is proportional to command speed value.
Speed (r/min)	This displays the actual rotation speed of motor.
Load (%)	This displays the motor load.
Max current 1 (%)	This displays the motor current command in terms of continuous current during stalling. An absolute value of the current command peak value sampled after the power ON is displayed.
Max current 2 (%)	This displays the motor current command in terms of continuous current during stalling. An absolute value of the current command peak value sampled in most recent 2 seconds is displayed.
Max current 3 (%)	This displays the FB value of the motor current in terms of continuous current during stalling. An absolute value of the current FB peak value sampled in most recent 2 seconds is displayed.
Overload (%)	This is the data used to monitor the motor overload.
Regen load (%)	This is the data used to monitor the resistance overload state when the resistance regenerative power supply is connected.
Est disturb torq (%)	This displays the estimated disturbance torque in terms of stall rated torque when the disturbance observer is valid.
Max disturb torq (%)	This displays the estimated disturbance torque in terms of stall rated torque when the collision detection function is adjusted. An absolute value of the estimated disturbance torque peak value sampled in most recent 2 seconds is displayed.
Load inertia R. (%)	This displays the estimated load inertia ratio when the collision detection function is adjusted.
Temperature (°C)	This displays the thermistor temperature.

Display items	Details
AFLT frequency (Hz)	This displays the current operation frequency of the adaptive filter.
AFLT gain (dB)	This displays the current filter depth of the adaptive filter.
LED display	This displays the 7-segment LED of the driver.
Alarm	This displays the alarms and warnings other than the LED display.
Cycle counter (p)	This displays the position within one rotation of the encoder detector. The position is displayed within one rotation in the range of "0" to "RNG (movement units) × 1000" using the grid point value as "0".
Grid space	This displays the grid space for the reference position return. (Command unit)
Grid amnt	This displays the distance from the dog-off point to the grid point when the dog-type reference position return is displayed. The grid mask amount is not included. (Command unit)
Machine posn	This displays the NC basic machine coordinate system position. (Command unit)
Motor end FB	This displays the feedback value of the speed detector. (Command unit)
FB error (i)	This displays the error of the motor end FB and machine end FB.
DFB compen amnt (i)	This displays the compensation pulse amount during dual feedback control.
Sync tap err (mm)	This displays the synchronous error width between the spindle and the drilling axis during the synchronous tapping. (mm) (Note 1) (When the parameter "#1041 I_Inch" is set to "1", "Sync tap err (inch)" is displayed.)
Sync tap err (deg)	This displays the synchronous error angle between the spindle and the drilling axis during the synchronous tapping. (degree) (Note 1)

(Note) Synchronous tapping error

This displays the maximum values of the synchronous tapping error that occur during the synchronous tapping.

The synchronous tapping error means the motor tracking delay for the commanded positions of the spindle and the tapping axis.

The positive synchronous tapping error means that the tapping axis is delayed responding to the spindle, and the negative synchronous tapping error means that the spindle is delayed responding to the tapping axis.

Data name	Details
Synchronous tapping error width (Max value)	<ul style="list-style-type: none"> This outputs the data of which absolute value is the largest among the synchronous tapping error width (-99999.999 to 99999.999 mm) during the synchronous tapping modal. This data will be initialized to "0" when entering the synchronous tapping modal or restoring the power. Other than that, the data continues to display the maximum value.
Synchronous tapping error angle (Max value)	<ul style="list-style-type: none"> This outputs the data of which absolute value is the largest among the synchronous tapping error angle (-99999.999 to 99999.999 °) during the synchronous tapping modal. This data will be initialized to "0" when entering the synchronous tapping modal or restoring the power. Other than that, the data continues to display the maximum value.

Display item	Details																																					
Control input 1L	This displays the control input signals from the NC.																																					
Control input 1H	<table border="1" data-bbox="576 338 1414 920"> <thead> <tr> <th></th> <th>Bit</th> <th>Details</th> </tr> </thead> <tbody> <tr> <td rowspan="8">1L</td> <td>0</td> <td>READY ON command</td> </tr> <tr> <td>1</td> <td>Servo ON command</td> </tr> <tr> <td>2</td> <td></td> </tr> <tr> <td>3</td> <td></td> </tr> <tr> <td>4</td> <td></td> </tr> <tr> <td>5</td> <td></td> </tr> <tr> <td>6</td> <td></td> </tr> <tr> <td>7</td> <td>Alarm reset command</td> </tr> <tr> <td rowspan="8">1H</td> <td>8</td> <td>Torque limit selection command 1</td> </tr> <tr> <td>9</td> <td>Torque limit selection command 2</td> </tr> <tr> <td>A</td> <td>Torque limit selection command 3</td> </tr> <tr> <td>B</td> <td></td> </tr> <tr> <td>C</td> <td></td> </tr> <tr> <td>D</td> <td></td> </tr> <tr> <td>E</td> <td></td> </tr> <tr> <td>F</td> <td></td> </tr> </tbody> </table>		Bit	Details	1L	0	READY ON command	1	Servo ON command	2		3		4		5		6		7	Alarm reset command	1H	8	Torque limit selection command 1	9	Torque limit selection command 2	A	Torque limit selection command 3	B		C		D		E		F	
	Bit	Details																																				
1L	0	READY ON command																																				
	1	Servo ON command																																				
	2																																					
	3																																					
	4																																					
	5																																					
	6																																					
	7	Alarm reset command																																				
1H	8	Torque limit selection command 1																																				
	9	Torque limit selection command 2																																				
	A	Torque limit selection command 3																																				
	B																																					
	C																																					
	D																																					
	E																																					
	F																																					
Control input 2L	This displays the control input signals from the NC.																																					
Control input 2H	<table border="1" data-bbox="576 1037 1414 1552"> <thead> <tr> <th></th> <th>Bit</th> <th>Details</th> </tr> </thead> <tbody> <tr> <td rowspan="8">2L</td> <td>0</td> <td></td> </tr> <tr> <td>1</td> <td></td> </tr> <tr> <td>2</td> <td></td> </tr> <tr> <td>3</td> <td></td> </tr> <tr> <td>4</td> <td></td> </tr> <tr> <td>5</td> <td></td> </tr> <tr> <td>6</td> <td></td> </tr> <tr> <td>7</td> <td></td> </tr> <tr> <td rowspan="8">2H</td> <td>8</td> <td></td> </tr> <tr> <td>9</td> <td>Speed observation command valid</td> </tr> <tr> <td>A</td> <td>Door closed (controller)</td> </tr> <tr> <td>B</td> <td>Door closed (all drive units)</td> </tr> <tr> <td>C</td> <td></td> </tr> <tr> <td>D</td> <td></td> </tr> <tr> <td>E</td> <td></td> </tr> <tr> <td>F</td> <td></td> </tr> </tbody> </table>		Bit	Details	2L	0		1		2		3		4		5		6		7		2H	8		9	Speed observation command valid	A	Door closed (controller)	B	Door closed (all drive units)	C		D		E		F	
	Bit	Details																																				
2L	0																																					
	1																																					
	2																																					
	3																																					
	4																																					
	5																																					
	6																																					
	7																																					
2H	8																																					
	9	Speed observation command valid																																				
	A	Door closed (controller)																																				
	B	Door closed (all drive units)																																				
	C																																					
	D																																					
	E																																					
	F																																					

Display item	Details																																					
Control input 3L	This displays the control input signals from the NC.																																					
Control input 3H	<table border="1"> <thead> <tr> <th data-bbox="576 338 651 367"></th> <th data-bbox="651 338 767 367">Bit</th> <th data-bbox="767 338 1410 367">Details</th> </tr> </thead> <tbody> <tr><td data-bbox="576 367 651 396" rowspan="8">3L</td><td data-bbox="651 367 767 396">0</td><td data-bbox="767 367 1410 396"></td></tr> <tr><td data-bbox="651 396 767 425">1</td><td data-bbox="767 396 1410 425"></td></tr> <tr><td data-bbox="651 425 767 454">2</td><td data-bbox="767 425 1410 454"></td></tr> <tr><td data-bbox="651 454 767 483">3</td><td data-bbox="767 454 1410 483"></td></tr> <tr><td data-bbox="651 483 767 512">4</td><td data-bbox="767 483 1410 512"></td></tr> <tr><td data-bbox="651 512 767 542">5</td><td data-bbox="767 512 1410 542"></td></tr> <tr><td data-bbox="651 542 767 571">6</td><td data-bbox="767 542 1410 571"></td></tr> <tr><td data-bbox="651 571 767 600">7</td><td data-bbox="767 571 1410 600"></td></tr> <tr><td data-bbox="576 600 651 629" rowspan="8">3H</td><td data-bbox="651 600 767 629">8</td><td data-bbox="767 600 1410 629"></td></tr> <tr><td data-bbox="651 629 767 658">9</td><td data-bbox="767 629 1410 658"></td></tr> <tr><td data-bbox="651 658 767 687">A</td><td data-bbox="767 658 1410 687"></td></tr> <tr><td data-bbox="651 687 767 716">B</td><td data-bbox="767 687 1410 716"></td></tr> <tr><td data-bbox="651 716 767 745">C</td><td data-bbox="767 716 1410 745"></td></tr> <tr><td data-bbox="651 745 767 775">D</td><td data-bbox="767 745 1410 775"></td></tr> <tr><td data-bbox="651 775 767 804">E</td><td data-bbox="767 775 1410 804"></td></tr> <tr><td data-bbox="651 804 767 833">F</td><td data-bbox="767 804 1410 833"></td></tr> </tbody> </table>		Bit	Details	3L	0		1		2		3		4		5		6		7		3H	8		9		A		B		C		D		E		F	
	Bit	Details																																				
3L	0																																					
	1																																					
	2																																					
	3																																					
	4																																					
	5																																					
	6																																					
	7																																					
3H	8																																					
	9																																					
	A																																					
	B																																					
	C																																					
	D																																					
	E																																					
	F																																					
Control input 4L	This displays the control input signals from the NC.																																					
Control input 4H	<table border="1"> <thead> <tr> <th data-bbox="576 969 651 999"></th> <th data-bbox="651 969 767 999">Bit</th> <th data-bbox="767 969 1410 999">Details</th> </tr> </thead> <tbody> <tr><td data-bbox="576 999 651 1028" rowspan="8">4L</td><td data-bbox="651 999 767 1028">0</td><td data-bbox="767 999 1410 1028"></td></tr> <tr><td data-bbox="651 1028 767 1057">1</td><td data-bbox="767 1028 1410 1057" rowspan="3">Spindle control mode selection command 1, 2, 3</td></tr> <tr><td data-bbox="651 1057 767 1086">2</td></tr> <tr><td data-bbox="651 1086 767 1115">3</td></tr> <tr><td data-bbox="651 1115 767 1144">4</td><td data-bbox="767 1115 1410 1144"></td></tr> <tr><td data-bbox="651 1144 767 1173">5</td><td data-bbox="767 1144 1410 1173">Gear selection command 1</td></tr> <tr><td data-bbox="651 1173 767 1202">6</td><td data-bbox="767 1173 1410 1202">Gear selection command 2</td></tr> <tr><td data-bbox="651 1202 767 1232">7</td><td data-bbox="767 1202 1410 1232"></td></tr> <tr><td data-bbox="576 1232 651 1261" rowspan="8">4H</td><td data-bbox="651 1232 767 1261">8</td><td data-bbox="767 1232 1410 1261"></td></tr> <tr><td data-bbox="651 1261 767 1290">9</td><td data-bbox="767 1261 1410 1290"></td></tr> <tr><td data-bbox="651 1290 767 1319">A</td><td data-bbox="767 1290 1410 1319"></td></tr> <tr><td data-bbox="651 1319 767 1348">B</td><td data-bbox="767 1319 1410 1348"></td></tr> <tr><td data-bbox="651 1348 767 1377">C</td><td data-bbox="767 1348 1410 1377">M-coil switch command</td></tr> <tr><td data-bbox="651 1377 767 1406">D</td><td data-bbox="767 1377 1410 1406">L-coil switch command</td></tr> <tr><td data-bbox="651 1406 767 1435">E</td><td data-bbox="767 1406 1410 1435">Sub-motor selection command</td></tr> <tr><td data-bbox="651 1435 767 1464">F</td><td data-bbox="767 1435 1410 1464"></td></tr> </tbody> </table>		Bit	Details	4L	0		1	Spindle control mode selection command 1, 2, 3	2	3	4		5	Gear selection command 1	6	Gear selection command 2	7		4H	8		9		A		B		C	M-coil switch command	D	L-coil switch command	E	Sub-motor selection command	F			
	Bit	Details																																				
4L	0																																					
	1	Spindle control mode selection command 1, 2, 3																																				
	2																																					
	3																																					
	4																																					
	5	Gear selection command 1																																				
	6	Gear selection command 2																																				
	7																																					
4H	8																																					
	9																																					
	A																																					
	B																																					
	C	M-coil switch command																																				
	D	L-coil switch command																																				
	E	Sub-motor selection command																																				
	F																																					

Display item	Details																																					
Control input 5L	This displays the control input signals from the NC.																																					
Control input 5H	<table border="1" data-bbox="576 338 1414 920"> <thead> <tr> <th></th> <th>Bit</th> <th>Details</th> </tr> </thead> <tbody> <tr><td rowspan="7">5L</td><td>0</td><td></td></tr> <tr><td>1</td><td></td></tr> <tr><td>2</td><td></td></tr> <tr><td>3</td><td></td></tr> <tr><td>4</td><td></td></tr> <tr><td>5</td><td></td></tr> <tr><td>6</td><td></td></tr> <tr><td rowspan="7">5H</td><td>7</td><td></td></tr> <tr><td>8</td><td></td></tr> <tr><td>9</td><td></td></tr> <tr><td>A</td><td></td></tr> <tr><td>B</td><td></td></tr> <tr><td>C</td><td></td></tr> <tr><td>D</td><td></td></tr> <tr><td>E</td><td>Spindle holding force up</td></tr> <tr><td>F</td><td></td></tr> </tbody> </table>		Bit	Details	5L	0		1		2		3		4		5		6		5H	7		8		9		A		B		C		D		E	Spindle holding force up	F	
	Bit	Details																																				
5L	0																																					
	1																																					
	2																																					
	3																																					
	4																																					
	5																																					
	6																																					
5H	7																																					
	8																																					
	9																																					
	A																																					
	B																																					
	C																																					
	D																																					
E	Spindle holding force up																																					
F																																						
Control input 6L	This displays the control input signals from the NC.																																					
Control input 6H	<table border="1" data-bbox="576 1032 1414 1552"> <thead> <tr> <th></th> <th>Bit</th> <th>Details</th> </tr> </thead> <tbody> <tr><td rowspan="7">6L</td><td>0</td><td></td></tr> <tr><td>1</td><td></td></tr> <tr><td>2</td><td></td></tr> <tr><td>3</td><td></td></tr> <tr><td>4</td><td></td></tr> <tr><td>5</td><td></td></tr> <tr><td>6</td><td></td></tr> <tr><td rowspan="7">6H</td><td>7</td><td></td></tr> <tr><td>8</td><td></td></tr> <tr><td>9</td><td></td></tr> <tr><td>A</td><td></td></tr> <tr><td>B</td><td></td></tr> <tr><td>C</td><td></td></tr> <tr><td>D</td><td></td></tr> <tr><td>E</td><td></td></tr> <tr><td>F</td><td></td></tr> </tbody> </table>		Bit	Details	6L	0		1		2		3		4		5		6		6H	7		8		9		A		B		C		D		E		F	
	Bit	Details																																				
6L	0																																					
	1																																					
	2																																					
	3																																					
	4																																					
	5																																					
	6																																					
6H	7																																					
	8																																					
	9																																					
	A																																					
	B																																					
	C																																					
	D																																					
E																																						
F																																						

Display item	Details																																					
Control output 1L	This displays the control output signals to the NC.																																					
Control output 1H	<table border="1" data-bbox="576 338 1415 920"> <thead> <tr> <th></th> <th>Bit</th> <th>Details</th> </tr> </thead> <tbody> <tr> <td rowspan="7">1L</td> <td>0</td> <td>In READY ON</td> </tr> <tr> <td>1</td> <td>In Servo ON</td> </tr> <tr> <td>2</td> <td></td> </tr> <tr> <td>3</td> <td></td> </tr> <tr> <td>4</td> <td></td> </tr> <tr> <td>5</td> <td></td> </tr> <tr> <td>6</td> <td></td> </tr> <tr> <td rowspan="7">1H</td> <td>7</td> <td>In alarm occurrence</td> </tr> <tr> <td>8</td> <td>In torque limit selection 1</td> </tr> <tr> <td>9</td> <td>In torque limit selection 2</td> </tr> <tr> <td>A</td> <td>In torque limit selection 3</td> </tr> <tr> <td>B</td> <td></td> </tr> <tr> <td>C</td> <td>In-position</td> </tr> <tr> <td>D</td> <td>In torque limitation</td> </tr> <tr> <td>E</td> <td></td> </tr> <tr> <td>F</td> <td>In warning occurrence</td> </tr> </tbody> </table>		Bit	Details	1L	0	In READY ON	1	In Servo ON	2		3		4		5		6		1H	7	In alarm occurrence	8	In torque limit selection 1	9	In torque limit selection 2	A	In torque limit selection 3	B		C	In-position	D	In torque limitation	E		F	In warning occurrence
	Bit	Details																																				
1L	0	In READY ON																																				
	1	In Servo ON																																				
	2																																					
	3																																					
	4																																					
	5																																					
	6																																					
1H	7	In alarm occurrence																																				
	8	In torque limit selection 1																																				
	9	In torque limit selection 2																																				
	A	In torque limit selection 3																																				
	B																																					
	C	In-position																																				
	D	In torque limitation																																				
E																																						
F	In warning occurrence																																					
Control output 2L	This displays the control output signals to the NC.																																					
Control output 2H	<table border="1" data-bbox="576 1034 1415 1550"> <thead> <tr> <th></th> <th>Bit</th> <th>Details</th> </tr> </thead> <tbody> <tr> <td rowspan="7">2L</td> <td>0</td> <td>Z-phase passed</td> </tr> <tr> <td>1</td> <td></td> </tr> <tr> <td>2</td> <td></td> </tr> <tr> <td>3</td> <td>In zero speed</td> </tr> <tr> <td>4</td> <td></td> </tr> <tr> <td>5</td> <td></td> </tr> <tr> <td>6</td> <td></td> </tr> <tr> <td rowspan="7">2H</td> <td>7</td> <td>In external emergency stop</td> </tr> <tr> <td>8</td> <td></td> </tr> <tr> <td>9</td> <td>In speed observation</td> </tr> <tr> <td>A</td> <td>Door closed (controller)</td> </tr> <tr> <td>B</td> <td>Door closed (local drive units)</td> </tr> <tr> <td>C</td> <td></td> </tr> <tr> <td>D</td> <td></td> </tr> <tr> <td>E</td> <td></td> </tr> <tr> <td>F</td> <td></td> </tr> </tbody> </table>		Bit	Details	2L	0	Z-phase passed	1		2		3	In zero speed	4		5		6		2H	7	In external emergency stop	8		9	In speed observation	A	Door closed (controller)	B	Door closed (local drive units)	C		D		E		F	
	Bit	Details																																				
2L	0	Z-phase passed																																				
	1																																					
	2																																					
	3	In zero speed																																				
	4																																					
	5																																					
	6																																					
2H	7	In external emergency stop																																				
	8																																					
	9	In speed observation																																				
	A	Door closed (controller)																																				
	B	Door closed (local drive units)																																				
	C																																					
	D																																					
E																																						
F																																						

Display item	Details																																					
Control output 3L	This displays the control output signals to the NC.																																					
Control output 3H	<table border="1" data-bbox="576 338 1414 846"> <thead> <tr> <th></th> <th>Bit</th> <th>Details</th> </tr> </thead> <tbody> <tr><td rowspan="7">3L</td><td>0</td><td></td></tr> <tr><td>1</td><td></td></tr> <tr><td>2</td><td></td></tr> <tr><td>3</td><td></td></tr> <tr><td>4</td><td></td></tr> <tr><td>5</td><td></td></tr> <tr><td>6</td><td></td></tr> <tr><td rowspan="7">3H</td><td>7</td><td></td></tr> <tr><td>8</td><td></td></tr> <tr><td>9</td><td></td></tr> <tr><td>A</td><td></td></tr> <tr><td>B</td><td></td></tr> <tr><td>C</td><td></td></tr> <tr><td>D</td><td></td></tr> <tr><td>E</td><td></td></tr> <tr><td>F</td><td></td></tr> </tbody> </table>		Bit	Details	3L	0		1		2		3		4		5		6		3H	7		8		9		A		B		C		D		E		F	
	Bit	Details																																				
3L	0																																					
	1																																					
	2																																					
	3																																					
	4																																					
	5																																					
	6																																					
3H	7																																					
	8																																					
	9																																					
	A																																					
	B																																					
	C																																					
	D																																					
E																																						
F																																						
Control output 4L	This displays the control output signals to the NC.																																					
Control output 4H	<table border="1" data-bbox="576 967 1414 1476"> <thead> <tr> <th></th> <th>Bit</th> <th>Details</th> </tr> </thead> <tbody> <tr><td rowspan="7">4L</td><td>0</td><td></td></tr> <tr><td>1</td><td rowspan="3">Spindle control mode selected 1, 2, 3</td></tr> <tr><td>2</td></tr> <tr><td>3</td></tr> <tr><td>4</td><td></td></tr> <tr><td>5</td><td>Gear selected 1</td></tr> <tr><td>6</td><td>Gear selected 2</td></tr> <tr><td rowspan="7">4H</td><td>7</td><td></td></tr> <tr><td>8</td><td></td></tr> <tr><td>9</td><td></td></tr> <tr><td>A</td><td></td></tr> <tr><td>B</td><td></td></tr> <tr><td>C</td><td>M-coil switched</td></tr> <tr><td>D</td><td>L- coil switched</td></tr> <tr><td>E</td><td>Sub-motor selected</td></tr> <tr><td>F</td><td></td></tr> </tbody> </table>		Bit	Details	4L	0		1	Spindle control mode selected 1, 2, 3	2	3	4		5	Gear selected 1	6	Gear selected 2	4H	7		8		9		A		B		C	M-coil switched	D	L- coil switched	E	Sub-motor selected	F			
	Bit	Details																																				
4L	0																																					
	1	Spindle control mode selected 1, 2, 3																																				
	2																																					
	3																																					
	4																																					
	5	Gear selected 1																																				
	6	Gear selected 2																																				
4H	7																																					
	8																																					
	9																																					
	A																																					
	B																																					
	C	M-coil switched																																				
	D	L- coil switched																																				
E	Sub-motor selected																																					
F																																						

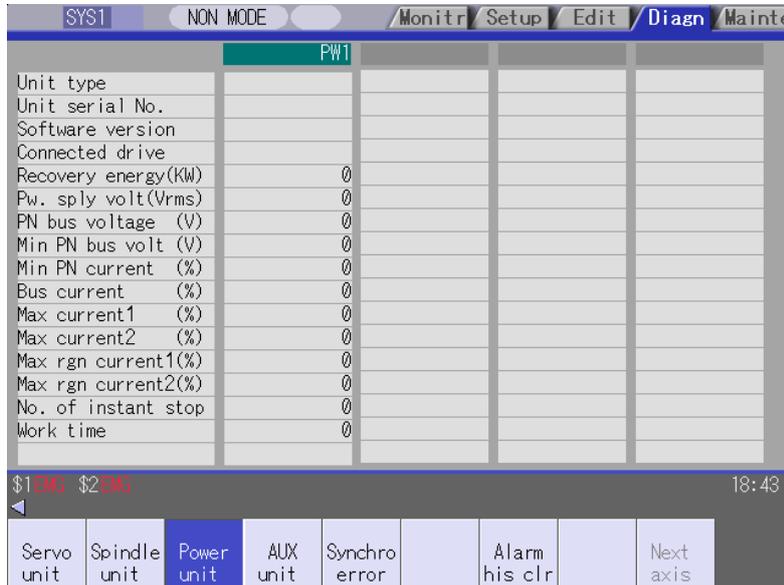
Display item	Details																																					
Control output 5L	This displays the control output signals to the NC.																																					
Control output 5H	<table border="1" data-bbox="576 338 1414 864"> <thead> <tr> <th></th> <th>Bit</th> <th>Details</th> </tr> </thead> <tbody> <tr> <td rowspan="8">5L</td> <td>0</td> <td>Current detection</td> </tr> <tr> <td>1</td> <td>Speed detection</td> </tr> <tr> <td>2</td> <td></td> </tr> <tr> <td>3</td> <td></td> </tr> <tr> <td>4</td> <td></td> </tr> <tr> <td>5</td> <td></td> </tr> <tr> <td>6</td> <td>In coil changeover</td> </tr> <tr> <td>7</td> <td></td> </tr> <tr> <td rowspan="7">5H</td> <td>8</td> <td>1-amplifire 2-motor switching</td> </tr> <tr> <td>9</td> <td>2nd speed detection</td> </tr> <tr> <td>A</td> <td></td> </tr> <tr> <td>B</td> <td></td> </tr> <tr> <td>C</td> <td></td> </tr> <tr> <td>D</td> <td></td> </tr> <tr> <td>E</td> <td>In spindle holding force up</td> </tr> <tr> <td>F</td> <td>2nd in-position</td> </tr> </tbody> </table>		Bit	Details	5L	0	Current detection	1	Speed detection	2		3		4		5		6	In coil changeover	7		5H	8	1-amplifire 2-motor switching	9	2nd speed detection	A		B		C		D		E	In spindle holding force up	F	2nd in-position
	Bit	Details																																				
5L	0	Current detection																																				
	1	Speed detection																																				
	2																																					
	3																																					
	4																																					
	5																																					
	6	In coil changeover																																				
	7																																					
5H	8	1-amplifire 2-motor switching																																				
	9	2nd speed detection																																				
	A																																					
	B																																					
	C																																					
	D																																					
	E	In spindle holding force up																																				
F	2nd in-position																																					
Control output 6L	This displays the control output signals to the NC.																																					
Control output 6H	<table border="1" data-bbox="576 981 1414 1496"> <thead> <tr> <th></th> <th>Bit</th> <th>Details</th> </tr> </thead> <tbody> <tr> <td rowspan="8">6L</td> <td>0</td> <td></td> </tr> <tr> <td>1</td> <td></td> </tr> <tr> <td>2</td> <td></td> </tr> <tr> <td>3</td> <td></td> </tr> <tr> <td>4</td> <td></td> </tr> <tr> <td>5</td> <td></td> </tr> <tr> <td>6</td> <td></td> </tr> <tr> <td>7</td> <td></td> </tr> <tr> <td rowspan="7">6H</td> <td>8</td> <td></td> </tr> <tr> <td>9</td> <td></td> </tr> <tr> <td>A</td> <td></td> </tr> <tr> <td>B</td> <td></td> </tr> <tr> <td>C</td> <td></td> </tr> <tr> <td>D</td> <td></td> </tr> <tr> <td>E</td> <td></td> </tr> <tr> <td>F</td> <td></td> </tr> </tbody> </table>		Bit	Details	6L	0		1		2		3		4		5		6		7		6H	8		9		A		B		C		D		E		F	
	Bit	Details																																				
6L	0																																					
	1																																					
	2																																					
	3																																					
	4																																					
	5																																					
	6																																					
	7																																					
6H	8																																					
	9																																					
	A																																					
	B																																					
	C																																					
	D																																					
	E																																					
F																																						

Display item	Details
Unit type	This displays the spindle type.
Unit serial No.	This displays the spindle serial No.
Software version	This displays the software No. and version on the spindle side.
Motor end detect No	This displays the motor end detector serial No.
Mach. end detect No	This displays the machine end detector serial No.
Work time	This displays the READY ON cumulative time. (Units: 1hr)
Alarm hist 1: Time 1: Alarm : 8: Time 8: Alarm	This displays servo alarms that occurred in latest order with the following formats. Time : Work time when the alarm occurred. Alarm No. : Number of the servo alarms that occurred.
Maint hist 1 to 4	This displays the maintenance dates. Year : One digit Month : 1 to 9, X (Oct.), Y (Nov.), Z (Dec.)
Maint status	This displays the maintenance status.

5.4.3 Display Items for the Power Supply Unit

The various data related to the power supply is monitored.

Change the display items using the  key and  key to refer to the data.



Display items

Display item	Details
Unit type	This displays the power supply unit type.
Unit serial No.	This displays the serial No. of the power supply unit.
Software version	This displays the software version.
Connected drive	This displays the I/F channel No. (mcp_no, smcp_no) of the drive unit connected to each power supply unit.
Recovery energy(KW)	This displays the regenerative power every two seconds. (0 to 999kW)
Pw. sply volt (Vrms)	This displays the effective value of the power supply voltage. (0 to 999Vrms)
PN bus voltage (V)	This displays PN bus voltage. (0 to 999V)
Min PN bus volt (V)	This displays the minimum PN bus voltage after the NC power ON. (0 to 999V)
Min PN current (%)	This displays the bus current when PN bus voltage is at minimum. (driving: +, regenerative: -) (0 to 999%)
Bus current (%)	This displays the bus current. (driving: +, regenerative: -) (0 to 999%)
Max current1 (%)	This displays the maximum driving current after the NC power ON. (0 to 999%)
Max current2 (%)	This displays the maximum driving current in most recent 2 seconds. (0 to 999%)
Max rgn current1(%)	This displays the maximum regenerative current after the NC power ON. (0 to 999%)
Max rgn current2(%)	This displays the maximum regenerative current in most recent 2 seconds. (0 to 999%)
No. of instant stop	This displays the number of instantaneous stop exceeding 1 cycle of the power. (0 to 9999 times)
Work time	This displays the READY ON cumulative time. (Units: 1hr)

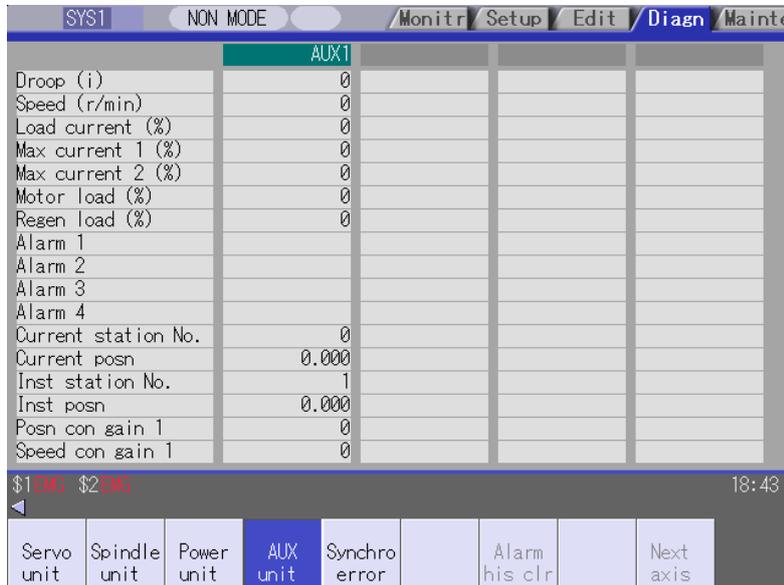
Display item	Details
Alarm hist 1: Time 1: Alarm : 8: Time 8: Alarm	This displays servo alarms that occurred in latest order with the following formats. Time : Work time when the alarm occurred Alarm No. : Number of the servo alarms that occurred
Maint hist 1 to 4	This displays the maintenance dates. Year : One digit Month : 1 to 9, X (Oct.), Y (Nov.), Z (Dec.)
Maint status	This displays the maintenance status.

5.4.4 Display Items for the Auxiliary Axis Unit [700 series only]

"Auxiliary axis unit" appears only when there is one or more valid auxiliary axis. (Base common parameter "#1044 auxno" is "1" or more).

The various data related to the auxiliary axis (MR-J2-CT) servo control is monitored.

For reference, change the display items using the  key and  key.



Data for the number of axes (up to six) set with the base common parameter "#1044 auxno (No. of connected MR-J2-CT units)" is displayed.

Display items

Display item	Details
Droop (i)	The error of the actual machine position to the commanded position is called droop. This difference is displayed.
Speed (r/min)	This displays the actual rotation speed of the motor.
Load current (%)	This displays the continuous executable load torque. This is 100% when the constant torque is generated.
Max current 1 (%)	This displays the commanded torque.
Max current 2 (%)	This displays the maximum generation torque of the commanded torque.
Motor load (%)	This displays the rate of the load torque to the tolerable load torque as a percentage.
Regen load (%)	This displays the rate of the regenerative power to the tolerable regenerative power as a percentage.
Alarm 1 to 4	This displays the system alarm, servo alarm, system warning, servo warning, operation alarm No. and alarm information.
Current station No.	This displays the number of the currently stopped station.
Current posn	This displays the coordinate of current position. (Unit: °)
Inst station No.	This displays the commanded station number for automatic operation, or the nearest station number to be stopped at for manual operation.
Inst posn	This displays the coordinate position corresponding to the target station number. (Unit: °)

Display item	Details
Posn con gain 1	This displays the position control gain 1.
Speed con gain 1	This displays the speed control gain 1.
Posn con gain 2	This displays the position control gain 2.
Speed con gain 2	This displays the speed control gain 2.
Speed int comp	This displays the speed integral compensation value.
Load inertia	This displays the estimated motor shaft conversion load inertia ratio value in respect to the motor's rotor inertia.
Unit type	This is the unit type.
Software version	This is the servo drive unit software version.
Motor type	This is the motor type.
Unit serial No.	This displays the unit serial No.
Alarm hist 1: No. 1: State : 6: No. 6: State	Alarm history : Alarm number Alarm history : Alarm details information

5.4.5 Display Items for the Synchronous Error

The "Synchronous error" appears only when the synchronous control axis option is valid.
The various data related to the synchronous error is monitored.

The screenshot shows a software interface for monitoring drive parameters. At the top, it displays 'SYS1', 'NON MODE', and navigation tabs: 'Monitr', 'Setup', 'Edit', 'Diagn', and 'Mainte'. The main display area is a table with columns for 'X1' and 'Y2'. The rows show the following data:

	X1	Y2
Slave axis	Y1	X2
Command error	0.000	0.000
FB error	0.000	0.000
FB error MAX1	0.000	0.000
FB error MAX2	0.000	0.000
Machine posn	0.000	0.000

At the bottom of the screen, there are status indicators '\$1EMG', '\$2EMG', and the time '18:44'. A row of buttons is visible: 'Servo unit', 'Spindle unit', 'Power unit', 'AUX unit', 'Synchro error' (highlighted), 'Alarm his clr', and 'Next axis'.

Display items

Display item	Details
Slave axis	This displays the slave axis name which is controlled following the master axis. The axis name corresponding to the axis No. set in the axis specification parameter "#1068 slavno (slave axis No.)" is displayed. The name set in the base axis specification parameter "#1022 axname2 (2nd axis name)" is displayed for the slave axis.
Command error	This is the deviation of the slave axis machine position in respect to the master axis. The error of the commanded position to the servo control section before pitch error compensation, relative position compensation and backlash compensation is displayed. If this error occurs, the parameters that should be the same for the synchronous axes are different. Command error = Command s - command m - Δ Command s : Slave axis commanded position Command m : Master axis commanded position Δ : Command s - command m at start of synchronous control
FB error	This is the deviation of the slave axis feedback position in respect to the feedback position from the master axis servomotor. The actual error of the machine position is displayed. The synchronous error check is carried out on this error. FB error = FBs - FBm - Δ FBs : Slave axis feedback position FBm : Master axis feedback position Δ : FBs - FBm at start of synchronous control
FB error MAX1	This displays the maximum FB error after the start of the synchronous control.
FB error MAX2	This displays the maximum FB error approx. every 30 seconds after the start of the synchronous control.
Machine posn	This displays the commanded machine position for the master axis.

5.4.6 Clearing the Alarm History

Operation method

- (1) Press the menu **Servo unit** or **Spindle unit**.
- (2) Using the menu **Next axis**, tab keys **←** and **→**, select the axis (device) from which to clear the alarm history.
- (3) Press the menu **Alarm his clr**.

→ The menu is highlighted, and a message appears to confirm the erasing. The alarm history1: Time appears at the head.

	X1	Y1	Z1	C1
Alarm hist 1:Time	1	1	0	0
1:Alarm	12	11	00	00
2:Time	2	1	0	0
2:Alarm	23	22	00	00
3:Time	1	2	0	0
3:Alarm	34	33	00	00
4:Time	2	2	0	0
4:Alarm	45	44	00	00
5:Time	0	3	0	0
5:Alarm	00	55	00	00
6:Time	0	0	0	0
6:Alarm	00	00	00	00
7:Time	0	0	0	0
7:Alarm	00	00	00	00
8:Time	0	0	0	0
8:Alarm	00	00	00	00
Maint hist 1				

Erased? (Y/N)

Servo unit Spindle unit Power unit AUX unit Synchro error Alarm his clr Next axis

- (4) Press the **Y** key.

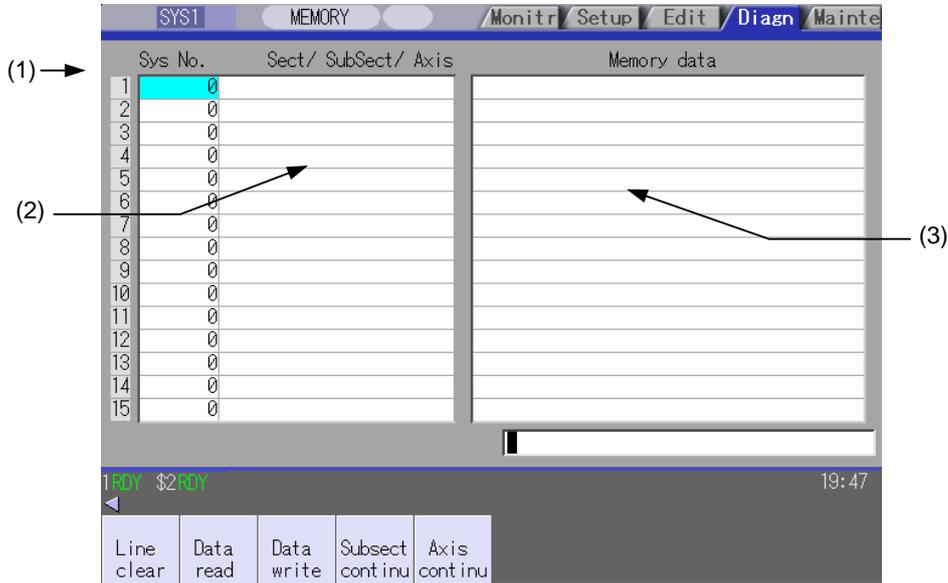
→ The alarm history data for the selected axis (device) is cleared to zero.

	X1	Y1	Z1	C1
Alarm hist 1:Time	1	0	0	0
1:Alarm	12	00	00	00
2:Time	2	0	0	0
2:Alarm	23	00	00	00
3:Time	1	0	0	0
3:Alarm	34	00	00	00
4:Time	2	0	0	0
4:Alarm	45	00	00	00
5:Time	0	0	0	0
5:Alarm	00	00	00	00
6:Time	0	0	0	0
6:Alarm	00	00	00	00
7:Time	0	0	0	0
7:Alarm	00	00	00	00
8:Time	0	0	0	0
8:Alarm	00	00	00	00
Maint hist 1				

Servo unit Spindle unit Power unit AUX unit Synchro error Alarm his clr Next axis

5.5 NC Memory Diagnosis Screen (NC Memory Diagn Screen)

The NC internal data can be displayed and rewritten on the screen. The custom API library's NC data read/write interface is used to display and rewrite the NC's internal data. The contents of the NC data can be displayed by designating the part system No., section No., sub-section No. and axis No. on this screen.



Display items

Display item	Details
(1) Index No.	This displays the registration No. of the NC memory data. When one of the "2. Data contents" is set, the number is highlighted indicating that the normal display of the data contents has stopped.
(2) Data contents	Part system No.: Designate the part system No. Designate "0" to designate the data common for the part systems. Section/sub-section/axis: Designate the section No., sub-section No. and axis No. of the data to be set and displayed. The setting format is, section No./sub-section No./axis No. (Note) The axis No. "1" is handled as the first axis. Designate "0" for the data which does not require an axis designation.
(3) Memory data	This displays the contents of the data.

5. Diagnosis Screens

5.5 NC Memory Diagnosis Screen (NC Memory Diagn Screen)

Menus

Menu	Details	Type	Reference
Line clear	This erases the information in the line where the cursor is. (One entire line becomes blank.) The cursor does not move at this time.	C	
Data read	The contents of the set address data (Part system No, Section/sub-section/axis) for all the lines are constantly displayed. The Index No. highlight (indicating data is being set) is released. The cursor appears in "Part system No" of that line.	C	5.5.1 Writing/Reading the Data Using the NC Data Designation
Data write	This writes the data (Note) in the setting area to the NC memory indicated by address data at the cursor position. The Data No. highlight (indicating data is being set) is released, and constant display is started. After writing, the cursor moves to "Part system No" of the next line.	A	
Subsect continu	Based on the data of the address data where the cursor is, this displays the continuous data to which the sub-section No. has been added to the address data from the line where the cursor is. The cursor moves to "Part system No " of that line.	C	
Axis continu	Based on the data of the address data where the cursor is, this displays the continuous data to which the axis No. has been added to the address data from the line where the cursor is. The cursor moves to "Part system No " of that line.	C	

(Note) Decimal, hexadecimal, floating point data and character string data writing is possible.
 Note that hexadecimal, floating point data and character strings may not be settable depending on the data.

Decimal : Integers without decimal points (Example) -1234
 Hexadecimal : An "H" is necessary at the end (Example) 1234H
 Floating point data : Data with a decimal point (Example) -12.3
 Character string data : Character string (Example) X

5.5.1 Writing/Reading the Data Using the NC Data Designation

When reading the Process parameter "#8007 Auto corner override", the following data is set.

(Example) Part system No. : 1
 Section No. : 126
 Sub-section No. : 8007
 Axis name : 0

Setting the data

- (1) Using the cursor keys, move the cursor to the "Part system No." position.  The cursor moves to the "Part system No." position.
- (2) Set the part system No.
 1  The index No. is highlighted, and the set value is displayed.
 The cursor moves to the right item position.
- (3) Separate the section No., sub-section No., axis No. with a "/", and set.
 126/8007/0  The set value appears.
 The cursor moves to the next "Sys No." line position.
- (4) Move the cursor to the set line position of "Sys No.", press the menu key .  The cursor moves to "memory data".
- (5) Set the data, and press the key.  Write processing is executed.
 The Index No. highlight returns to normal.

Reading the data

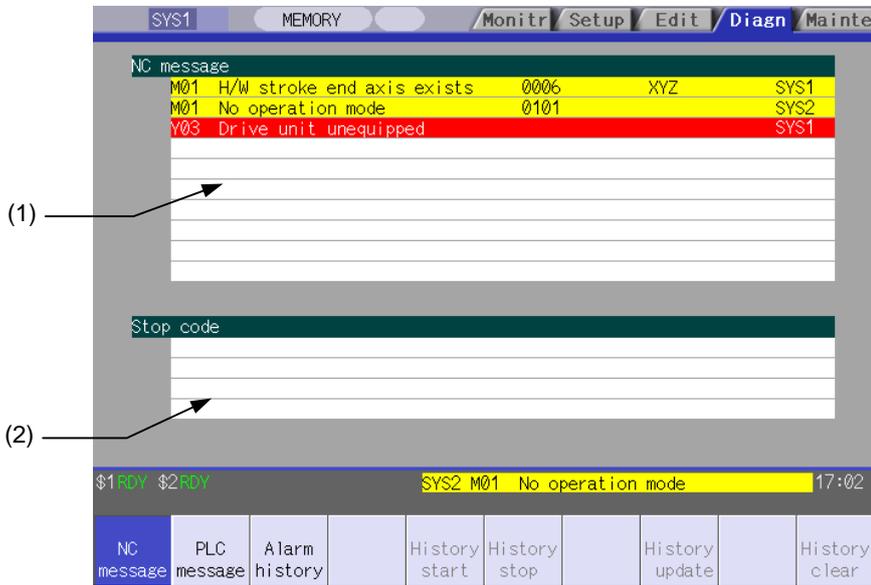
- (1) Set the part system No., section No., sub-section No., axis No. in the same way as step "Setting the data".
- (2) Press the menu key .  The index No. highlight is released, and the normal display of the memory data starts.
 The data format (data size, display format) of the data displayed in the memory data differs according to the data type to be displayed.

(Note) The cursor is constantly displayed. Using the cursor keys, the cursor can be moved to the part system No. area, section/sub-section/axis area.

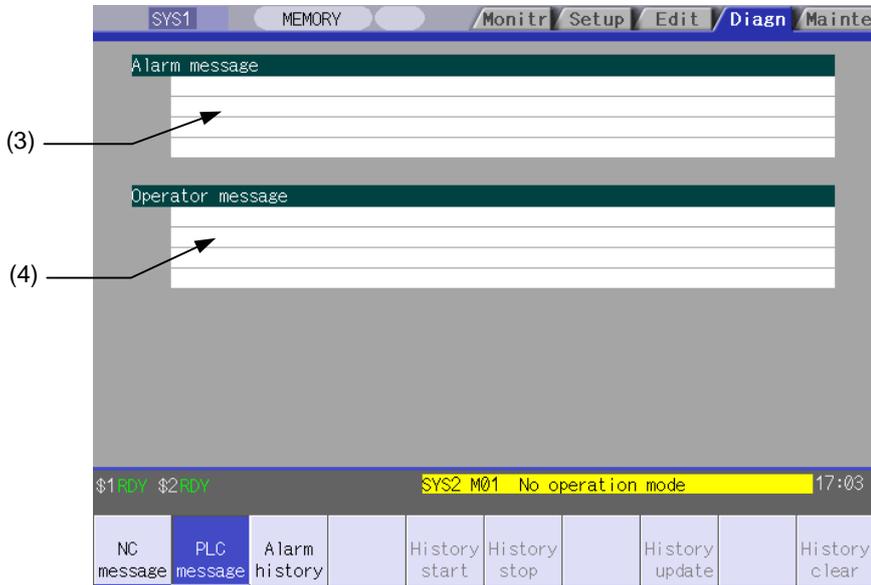
5.6 Alarm Screen

A list of currently occurring alarms or messages can be displayed on this screen. The displayed messages include the NC alarms, stop codes, alarm messages, operator messages, etc. A history of alarm information can be displayed also.

■ NC message



■ PLC message



Display items

Display item	Details
(1) NC alarm	This displays the operation alarms, program errors, MCP alarms, servo alarms, system alarms, etc. Up to 10 messages are displayed in order of priority.
(2) Stop code	This displays the automatic operation status or stop status during automatic operation. Up to 4 messages are displayed in order of priority.
(3) Alarm message	Using the PLC programs, this displays messages such as details of machine abnormalities. Up to 4 messages are displayed.
(4) Operator message	Using the PLC programs, this displays operator information messages. Macro alarm messages are also displayed in this field. Up to 4 messages are displayed.

Message display colors

The messages are color-coded in the following manner.

Message type		Character color	Background color
NC message	Alarm	White	Red
	Warning	Black	Yellow
Stop code		Black	Yellow
Alarm message		White	Red
Operator message		Black	Yellow

Axis name display

The axis name is displayed in messages for each axis. The axis name is displayed as shown below according to the axis type.

Axis type	Axis name display	Display example	Remarks
NC axis	Control axis name (Name of axis in part system)	XYZ	If the same message occurs for each part system, several NC axes are displayed together.
Spindle	'S' + spindle No.	S1S2	If the same message occurs, several spindles are displayed together.
PLC axis	'P' + PLC axis No.	P1P2	If the same message occurs, several PLC axes are displayed together.
Auxiliary axis	'A' + auxiliary axis No.	A1A2	If the same message occurs, several auxiliary axes are displayed together.

If the same message occurs for different axis types, they will appear as separate messages.

Part system display

The part system name is also displayed if the message is output for each part system. The part system name set in "#1169 system name" is displayed. The part system name does not appear for the 1-part system.

5.6.1 Alarm History

When an alarm occurs, the alarm information is recorded. When the NC power is ON, an alarm is automatically recorded in alarm history. Alarm information is recorded from the latest alarm to 512. Alarm information recorded in the history is NC message and a stop code displayed on "NC message" screen and alarm messages displayed on "PLC message" screen. The range etc. of record are shown as follows.

- Record condition : When an alarm occurs (When two or more alarms occur at the same time, up to five alarms are recorded.)
With multi-part system, 1st part system is given priority and recorded.
(Following 2nd part system, 3rd part system...)
- Number of history : 512 alarms (Whole)
- Range of record : NC alarm (alarm, warning), stop code, PLC alarm message

SYS1		MEMORY		Monitr	Setup	Edit	Diagn	Mainte
Alarm history								Page 1
0221	16:59:07	M01	No operation mode	0101				SYS2
0221	16:58:30	M01	H/W stroke end axis exists	0006	XYZ			SYS1
0221	16:58:30	N005	Invalid net communication	-103				SYS1
0221	16:58:30	M01	H/W stroke end axis exists	0006	C			SYS2
0221	16:52:46	M01	No operation mode	0101				SYS2
0221	16:52:09	M01	H/W stroke end axis exists	0006	XYZ			SYS1
0221	16:52:09	N005	Invalid net communication	-103				SYS1
0221	16:52:09	M01	H/W stroke end axis exists	0006	C			SYS2
0221	16:46:11	T01	H/W stroke end axis exists	0105				SYS1
0221	16:44:49	M01	No operation mode	0101				SYS2
0221	16:44:13	M01	H/W stroke end axis exists	0006	XYZ			SYS1
0221	16:44:13	N005	Invalid net communication	-103				SYS1
0221	16:44:13	M01	H/W stroke end axis exists	0006	C			SYS2
0221	16:42:06	M01	H/W stroke end axis exists	0006	XYZ			SYS1
0221	16:42:06	N005	Invalid net communication	-103				SYS1
0221	15:13:12	M01	No operation mode	0101				SYS2
\$1 --- \$2 --- SYS2 M01 No operation mode								17:03
NC message	PLC message	Alarm history		History start	History stop		History update	History clear

Menus

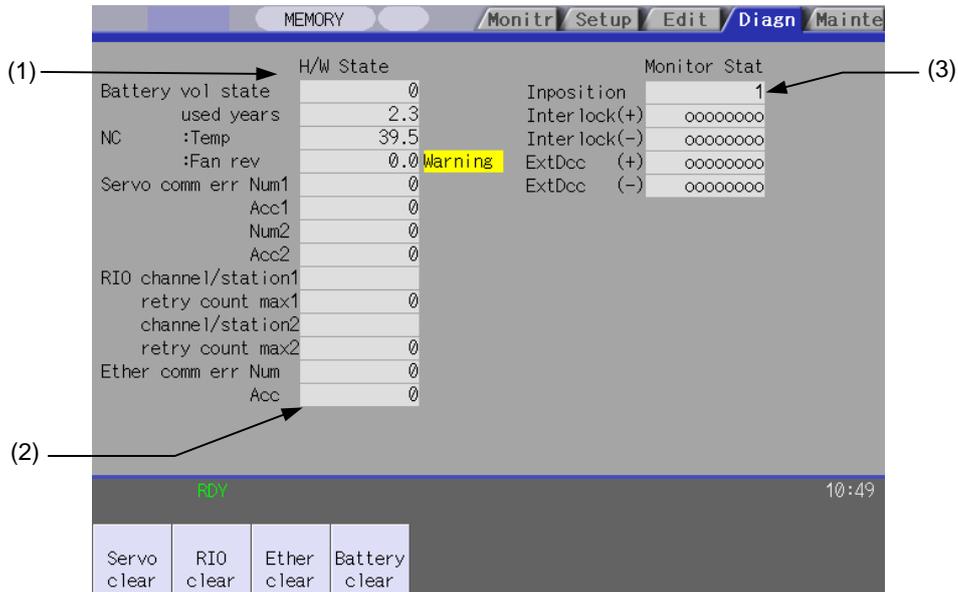
Menu	Details	Type	Reference
Alarm history	This displays the first page of the alarm history. The history sequentially displays 16 alarms per page from the latest alarm. If there are two or more NC alarms of same day and time, the alarms are sequentially displayed from the alarm and warning of 1st part system. Page is changed by  , and older histories are displayed. Page is changed by  , and newer histories are displayed.	B	
History start	The data collection of the alarm history is started. The operation message "The collection begin? (Y/N)" appears. When the Y or <input type="text" value="INPUT"/> key is pressed, the data collection is started after the operation message "The collection begin" appears. Press the N or other than <input type="text" value="INPUT"/> key when the data collection will be not started/restarted.	C	
History stop	The data collection of the alarm history is stopped. The operation message "The collection stop? (Y/N)" appears. When the Y or <input type="text" value="INPUT"/> key is pressed, the data collection is started after the operation message " The collection stop" appears. Press the N or other than <input type="text" value="INPUT"/> key when the data collection will be not stopped.	C	
History update	The alarm information of history is updated. When the history is updated, the page with latest alarm information (first page) is displayed. The history is updated even if changing to another screen, and returning to the alarm history screen.	C	
History clear	The alarm information of history is cleared. The operation message "Execute the collection data clear?(Y/N)" appears. When the Y or <input type="text" value="INPUT"/> key is pressed, the alarm information of history is cleared after the operation message "Data clear complete" appears. The first page is displayed when the history is cleared.	C	

(Note 1) History start, History stop, History update and History clear menus are valid when the alarm history function is valid and Alarm history menu is selected.

(Note 2) When the alarm history function is used for the first time, clear the alarm history contents by pressing menu. Unnecessary data may be recorded in the alarm history.

5.7 Self Diagnosis Screen

The H/W state and NC operation state can be confirmed on this screen.



Display items

Display item	Details																																	
(1) H/W state (common for part systems)	This displays H/W state of NC unit and display unit. As for the NC unit, the contents are as follows.																																	
	<table border="1"> <thead> <tr> <th>Display item</th> <th>Details</th> </tr> </thead> <tbody> <tr> <td colspan="2">NC</td> </tr> <tr> <td rowspan="4">Battery vol state</td> <td>This displays the current state of the battery voltage as 0 to 3 below.</td> </tr> <tr> <td>Condition</td> <td>Classification</td> </tr> <tr> <td>0 (normal state)</td> <td>--</td> </tr> <tr> <td>1 (battery drop)</td> <td>Cautions (gray)</td> </tr> <tr> <td rowspan="3">used years</td> <td>This displays approximate time of the battery used from the last replacement.</td> </tr> <tr> <td>Condition</td> <td>Classification</td> </tr> <tr> <td>Recommended battery use (5 years) ≤ Time for the battery used</td> <td>Warning (yellow)</td> </tr> <tr> <td rowspan="4">NC :Temp</td> <td>This displays the current temperature of the control unit.</td> </tr> <tr> <td>Condition</td> <td>Classification</td> </tr> <tr> <td>-8°C < Control unit temp. ≤ -3°C</td> <td>Cautions (gray)</td> </tr> <tr> <td>63°C ≤ Control unit temp. < 68°C</td> <td rowspan="2">Warning (yellow)</td> </tr> <tr> <td>Control unit temp. ≤ -8°C</td> </tr> <tr> <td rowspan="3">:Fan rev</td> <td>This displays the current fan rotation speed of the control unit.</td> </tr> <tr> <td>Condition</td> <td>Classification</td> </tr> <tr> <td>Fan rot. speed ≤ 4000 r/min</td> <td>Warning (yellow)</td> </tr> </tbody> </table>	Display item	Details	NC		Battery vol state	This displays the current state of the battery voltage as 0 to 3 below.	Condition	Classification	0 (normal state)	--	1 (battery drop)	Cautions (gray)	used years	This displays approximate time of the battery used from the last replacement.	Condition	Classification	Recommended battery use (5 years) ≤ Time for the battery used	Warning (yellow)	NC :Temp	This displays the current temperature of the control unit.	Condition	Classification	-8°C < Control unit temp. ≤ -3°C	Cautions (gray)	63°C ≤ Control unit temp. < 68°C	Warning (yellow)	Control unit temp. ≤ -8°C	:Fan rev	This displays the current fan rotation speed of the control unit.	Condition	Classification	Fan rot. speed ≤ 4000 r/min	Warning (yellow)
	Display item	Details																																
	NC																																	
	Battery vol state	This displays the current state of the battery voltage as 0 to 3 below.																																
		Condition	Classification																															
		0 (normal state)	--																															
		1 (battery drop)	Cautions (gray)																															
	used years	This displays approximate time of the battery used from the last replacement.																																
		Condition	Classification																															
		Recommended battery use (5 years) ≤ Time for the battery used	Warning (yellow)																															
	NC :Temp	This displays the current temperature of the control unit.																																
		Condition	Classification																															
		-8°C < Control unit temp. ≤ -3°C	Cautions (gray)																															
		63°C ≤ Control unit temp. < 68°C	Warning (yellow)																															
Control unit temp. ≤ -8°C																																		
:Fan rev	This displays the current fan rotation speed of the control unit.																																	
	Condition	Classification																																
	Fan rot. speed ≤ 4000 r/min	Warning (yellow)																																
(Continues to the next page)																																		

Display item	Details	
(Continued from the previous page)	Communication between NC unit and display unit	
	Servo comm err Num1	This displays the count of occurrence for "Y02 SV commu er: Recv frame No. 0051 xx04" after the power ON.
	Acc1	This displays the cumulated count of occurrence for " Y02 SV commu er: Recv frame 0051 xx04". Press the Servo clear menu to clear the cumulated count to "0".
	Servo comm err Num2	This displays the count of occurrence for "Y02 SV commu er: Data ID error 0051 xx03" after the power ON.
	Acc2	This displays the cumulated count of occurrence for "Y02 SV commu er: Data ID error 0051 xx03". Press the Servo clear menu to clear the cumulated count to "0".
	RIO channel/station1	This displays the Channel No./Station No. of occurrence for continuous error after the power ON.
	retry count max1	This displays the maximum value of the continuous error after the power ON.
	RIO channel/station2	This displays the Channel No./Station No. held even if the power OFF. Press the RIO clear menu to clear the Channel No./Station No. to "0/0".
	retry count max2	This displays the count held even if the power OFF. Press the RIO clear menu to clear the count to "0".
	Ether comm err Num	This displays the number of Ethernet communication error after PLC program is executed once.
	Acc	This displays the cumulated count of occurrence for " Ether communication error ". Press the Ether clear menu to clear the cumulated count to "0".

Display item	Details												
<p>(2) Operation state (Depends on part system)</p>	<p>This displays the state when the operation seems to be stopped in spite that the alarm does not occur. The following state can be confirmed.</p> <table border="1" data-bbox="566 369 1436 1366"> <thead> <tr> <th data-bbox="566 369 750 398">State</th> <th data-bbox="750 369 1436 398">Details</th> </tr> </thead> <tbody> <tr> <td data-bbox="566 398 750 510">In-position</td> <td data-bbox="750 398 1436 510"> <p>This displays "1" (in-position state) when the following conditions are satisfied for even one axis.</p> <ul style="list-style-type: none"> • No acceleration/deceleration delay for all axes • Within the in-position width set in the parameter for all axes </td> </tr> <tr> <td data-bbox="566 510 750 772">Interlock(+)</td> <td data-bbox="750 510 1436 772"> <p>When the auto interlock +n-th axis signal or the manual interlock +n-th axis signal is OFF, "1" appears for the n-th axis. (Explanation of the display)</p> <p style="text-align: center;">o o o o o o 1 o</p> <p style="text-align: center;">↑ ↑</p> <p style="text-align: center;">8th axis ... 1st axis</p> <p>In the above case, the 2nd axis is interlocked. Even when the number of usable axes is less than 8 in 1 part system, this displays 8 axes fixed.</p> </td> </tr> <tr> <td data-bbox="566 772 750 884">Interlock(-)</td> <td data-bbox="750 772 1436 884"> <p>When the auto interlock -n-th axis signal or the manual interlock -n-th axis signal is OFF, "1" appears for the n-th axis. The explanation of the display is same as for the "Interlock (+)".</p> </td> </tr> <tr> <td data-bbox="566 884 750 1227">ExtDcc (+)</td> <td data-bbox="750 884 1436 1227"> <p>When the control axis is moving in (+) direction, "1" appears for the axis if the external deceleration speed is valid, and the feedrate is clamped, exceeding the set value of the external deceleration speed. (Explanation of the display)</p> <p style="text-align: center;">o o o o o 1 o 1</p> <p style="text-align: center;">↑ ↑</p> <p style="text-align: center;">8th axis ... 1st axis</p> <p>In the above case, the 1st axis and the 3rd axis are in external deceleration speed. Even when the number of usable axes is less than 8 in 1 part system, this displays 8 axes fixed.</p> </td> </tr> <tr> <td data-bbox="566 1227 750 1366">ExtDcc (-)</td> <td data-bbox="750 1227 1436 1366"> <p>When the control axis is moving in (-) direction, "1" appears for the axis if the external deceleration speed is valid, and the feedrate is clamped, exceeding the set value of the external deceleration speed. The explanation of the display is same as for the "ExtDcc" (+).</p> </td> </tr> </tbody> </table>	State	Details	In-position	<p>This displays "1" (in-position state) when the following conditions are satisfied for even one axis.</p> <ul style="list-style-type: none"> • No acceleration/deceleration delay for all axes • Within the in-position width set in the parameter for all axes 	Interlock(+)	<p>When the auto interlock +n-th axis signal or the manual interlock +n-th axis signal is OFF, "1" appears for the n-th axis. (Explanation of the display)</p> <p style="text-align: center;">o o o o o o 1 o</p> <p style="text-align: center;">↑ ↑</p> <p style="text-align: center;">8th axis ... 1st axis</p> <p>In the above case, the 2nd axis is interlocked. Even when the number of usable axes is less than 8 in 1 part system, this displays 8 axes fixed.</p>	Interlock(-)	<p>When the auto interlock -n-th axis signal or the manual interlock -n-th axis signal is OFF, "1" appears for the n-th axis. The explanation of the display is same as for the "Interlock (+)".</p>	ExtDcc (+)	<p>When the control axis is moving in (+) direction, "1" appears for the axis if the external deceleration speed is valid, and the feedrate is clamped, exceeding the set value of the external deceleration speed. (Explanation of the display)</p> <p style="text-align: center;">o o o o o 1 o 1</p> <p style="text-align: center;">↑ ↑</p> <p style="text-align: center;">8th axis ... 1st axis</p> <p>In the above case, the 1st axis and the 3rd axis are in external deceleration speed. Even when the number of usable axes is less than 8 in 1 part system, this displays 8 axes fixed.</p>	ExtDcc (-)	<p>When the control axis is moving in (-) direction, "1" appears for the axis if the external deceleration speed is valid, and the feedrate is clamped, exceeding the set value of the external deceleration speed. The explanation of the display is same as for the "ExtDcc" (+).</p>
State	Details												
In-position	<p>This displays "1" (in-position state) when the following conditions are satisfied for even one axis.</p> <ul style="list-style-type: none"> • No acceleration/deceleration delay for all axes • Within the in-position width set in the parameter for all axes 												
Interlock(+)	<p>When the auto interlock +n-th axis signal or the manual interlock +n-th axis signal is OFF, "1" appears for the n-th axis. (Explanation of the display)</p> <p style="text-align: center;">o o o o o o 1 o</p> <p style="text-align: center;">↑ ↑</p> <p style="text-align: center;">8th axis ... 1st axis</p> <p>In the above case, the 2nd axis is interlocked. Even when the number of usable axes is less than 8 in 1 part system, this displays 8 axes fixed.</p>												
Interlock(-)	<p>When the auto interlock -n-th axis signal or the manual interlock -n-th axis signal is OFF, "1" appears for the n-th axis. The explanation of the display is same as for the "Interlock (+)".</p>												
ExtDcc (+)	<p>When the control axis is moving in (+) direction, "1" appears for the axis if the external deceleration speed is valid, and the feedrate is clamped, exceeding the set value of the external deceleration speed. (Explanation of the display)</p> <p style="text-align: center;">o o o o o 1 o 1</p> <p style="text-align: center;">↑ ↑</p> <p style="text-align: center;">8th axis ... 1st axis</p> <p>In the above case, the 1st axis and the 3rd axis are in external deceleration speed. Even when the number of usable axes is less than 8 in 1 part system, this displays 8 axes fixed.</p>												
ExtDcc (-)	<p>When the control axis is moving in (-) direction, "1" appears for the axis if the external deceleration speed is valid, and the feedrate is clamped, exceeding the set value of the external deceleration speed. The explanation of the display is same as for the "ExtDcc" (+).</p>												
<p>(3) Credit system</p>	<p>This displays the valid term by the credit system when the credit system is valid. This does not display when the credit system specification is invalid.</p>												

Menus

Menu	Details	Type	Reference
Servo clear	This clears the cumulated count of the servo communication error 1 and 2 to "0".	A	Clearing the cumulated counter to zero
RIO clear	This clears the cumulated count of the RIO communication error to "0".	A	
Ether clear	This clears the cumulated count of the Ether communication error to "0".	A	
Battery clear	This clears the time the battery has been used to "0".	A	

Clearing the cumulated counter to zero

(Example) Clearing the cumulated count of the servo communication error

- (1) Press the menu . ➔ A confirmation message will be displayed.
- (2) Press or . ➔ The cumulated count of the servo communication error 1 and 2 will be cleared to "0".
When other keys are pressed, it will not clear to "0".

This also applies to , and menu.

When using the multi-part system specification, switch the displayed part system by menu or the part system switching key .

5.8 Data Sampling Screen

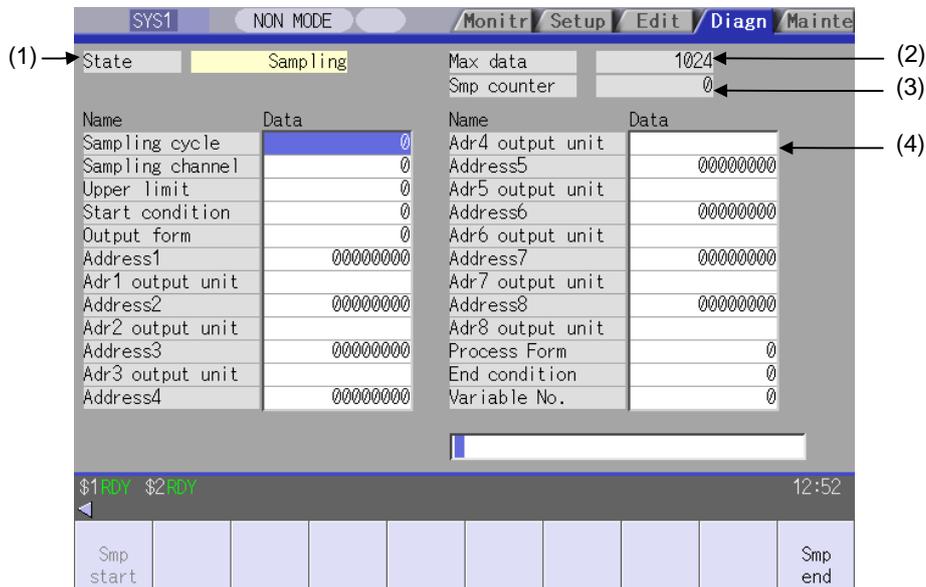
Sampling start/stop, sampling state display, setting the sampling parameters necessary for sampling are performed in the data sampling screen. The NC internal data (speed output from NC to the drive unit, or feedback data from the drive unit, etc.) can be sampled.

Also, the sampling data can be output externally on the I/O screen under the [Mainte] tag.

Sampling specifications

- Sampling cycle : 1.7ms * magnification
- Number of sampled axes : 1 to 20 axes
(Servo axis 1 to 16 + Spindle 1 to 4)
- Number of sampling channels : 1 to 8 points
- Number of sampled data items : Max. 1,310,720 points

(Note 1) The maximum number of sampled data items is the total number of data. When the number of sampled channels is increased, the number of data per channel will decrease.



Display items

Display item	Details								
(1) State	<p>This displays the current sampling status. (The displays, "Sampling" and "Trigger wait" are highlighted.)</p> <table border="1"> <thead> <tr> <th>Display</th> <th>State</th> </tr> </thead> <tbody> <tr> <td>Sampling</td> <td>Sampling is being executed.</td> </tr> <tr> <td>Sampling stop</td> <td>Sampling is not being executed, or the sampling process has completed.</td> </tr> <tr> <td>Trigger wait</td> <td>When the "Start condition" is not set to "0" (manual start), this indicates a state waiting for the sampling to start, detecting the trigger, after the menu Samp start is pressed.</td> </tr> </tbody> </table>	Display	State	Sampling	Sampling is being executed.	Sampling stop	Sampling is not being executed, or the sampling process has completed.	Trigger wait	When the "Start condition" is not set to "0" (manual start), this indicates a state waiting for the sampling to start, detecting the trigger, after the menu Samp start is pressed.
Display	State								
Sampling	Sampling is being executed.								
Sampling stop	Sampling is not being executed, or the sampling process has completed.								
Trigger wait	When the "Start condition" is not set to "0" (manual start), this indicates a state waiting for the sampling to start, detecting the trigger, after the menu Samp start is pressed.								

Display item	Details						
(2) Maximum data	This displays the maximum number of data possible for sampling. Normally, the number of data calculated from the setting value of "Upper limit" is displayed. The maximum number is 50% of the DRAM available memory.						
(3) Sampling counter	<p>This displays the position of the sampling buffer during the sampling process. When the sampling buffer is valid, the head of the buffer can be seen with the value of the counter at the end of sampling.</p> <p>Sampling buffer</p> <table border="1" style="display: inline-table; vertical-align: middle;"> <tr><td>0</td></tr> <tr><td>1</td></tr> <tr><td>:</td></tr> <tr><td>n-1</td></tr> <tr><td>n</td></tr> <tr><td>:</td></tr> </table> <p>Sampling counter (n)→</p> <p>End of buffer (newest data) Head of buffer (oldest data)</p>	0	1	:	n-1	n	:
0							
1							
:							
n-1							
n							
:							
(4) Sampling parameter	This displays and sets the parameters necessary for sampling.						

Menus

Menu	Details	Type	Reference
Samp start	<p>This starts sampling. This menu cannot be selected under the "Sampling" state. When the trigger is set (the "Start condition" is not set to "0"), the state will be "Trigger wait" until the trigger is detected. The sampling will not stop even if transferred to another screen during sampling.</p> <p>If this menu is pressed when the parameter "#1224 aux08/bit0" is set to "0", the operation message "Can't start sampling" is displayed, and the operation will not start.</p>	C	/
Samp end	<p>This stops sampling, and the state will be "sampling stop". The menu Samp start can be selected.</p>	C	

Sampling parameter

Parameter	Details	Setting range (unit)
Sampling cycle	<p>Set the sampling cycle.</p> <p>Cycle = 1.7ms * setting value</p> <p>(Example) When the setting value is 1: 1.7ms cycle When the setting value is 2: 3.5ms cycle</p>	1 to 255
Sampling channel	<p>Set the number of sampling channel.</p> <p>(Note) The sampling buffer will be divided by the set number of channels. When the number of sampling channels is increased, the number of data per channel will decrease.</p>	1 to 8
Upper limit	<p>Set the capacity of the buffer to be used for sampling.</p> <p>Buffer capacity = (setting value + 1) * 1024 items</p> <p>(Note) When the setting value is 0: 1024 items When the setting value is 1: 2048 items</p>	0 to 1279

Parameter	Details	Setting range (unit)																														
Start condition	<p>Select the condition for starting sampling.</p> <table border="1"> <tr> <td>0</td> <td>Manual start Sampling starts when <input type="text" value="Smp start"/> menu is pressed.</td> </tr> <tr> <td>1</td> <td>Variable No. Sampling starts when the variable set in "Variable No." is 0 or a value other than null. (Note 2)</td> </tr> <tr> <td>2</td> <td>PLC device Sampling starts at the rising edge of the signal set in "PLC device". (Note3)</td> </tr> <tr> <td>3</td> <td>Address condition, true Sampling starts when "Address", "Data" and "Data mask" conditions are true. (Note1)</td> </tr> <tr> <td>4</td> <td>Address condition, false Sampling starts when "Address", "Data" and "Data mask" conditions are false. (Note1)</td> </tr> </table>	0	Manual start Sampling starts when <input type="text" value="Smp start"/> menu is pressed.	1	Variable No. Sampling starts when the variable set in "Variable No." is 0 or a value other than null. (Note 2)	2	PLC device Sampling starts at the rising edge of the signal set in "PLC device". (Note3)	3	Address condition, true Sampling starts when "Address", "Data" and "Data mask" conditions are true. (Note1)	4	Address condition, false Sampling starts when "Address", "Data" and "Data mask" conditions are false. (Note1)	0 to 4																				
0	Manual start Sampling starts when <input type="text" value="Smp start"/> menu is pressed.																															
1	Variable No. Sampling starts when the variable set in "Variable No." is 0 or a value other than null. (Note 2)																															
2	PLC device Sampling starts at the rising edge of the signal set in "PLC device". (Note3)																															
3	Address condition, true Sampling starts when "Address", "Data" and "Data mask" conditions are true. (Note1)																															
4	Address condition, false Sampling starts when "Address", "Data" and "Data mask" conditions are false. (Note1)																															
Output form	<p>Select the data format when outputting the sampling data in text style.</p> <table border="1"> <tr> <td>0</td> <td>Outputs in decimal figure</td> </tr> <tr> <td>1</td> <td>Outputs in hexadecimal figure with 8 digits</td> </tr> </table> <p>(Note 1) When "Output form" is set to "1", and "#1004 ctrlunit" is "E (1nano)", the data larger than 1m will not be output correctly. When the output data exceeds 1m, the low-order 32bits of the sampling data will be output.</p> <p>(Note 2) Refer to the section "5.8.2 Outputting sampling data" for details on sampling data output.</p>	0	Outputs in decimal figure	1	Outputs in hexadecimal figure with 8 digits	0, 1																										
0	Outputs in decimal figure																															
1	Outputs in hexadecimal figure with 8 digits																															
Address1 : Address8	<p>Set the sampling address.</p> <p>(a) Index No. method The index No. is fixed regardless of the axis configuration. The low-order six digits of the address are explained below. (If the high-order two digits are not set, the setting will be interpreted as 00.)</p> <p>Setting range (Index No.)</p> <table border="1"> <thead> <tr> <th>Servo axis</th> <th>1stAX</th> <th>2ndAX</th> <th>...</th> <th>16thAX</th> </tr> </thead> <tbody> <tr> <td>Feedback position</td> <td>000100</td> <td>000200</td> <td>...</td> <td>000F00</td> </tr> <tr> <td>Commanded position</td> <td>000101</td> <td>000201</td> <td>...</td> <td>000F01</td> </tr> </tbody> </table> <table border="1"> <thead> <tr> <th>Spindle</th> <th>1stSP</th> <th>2ndSP</th> <th>...</th> <th>4thSP</th> </tr> </thead> <tbody> <tr> <td>Feedback position</td> <td>010000</td> <td>020000</td> <td>...</td> <td>040000</td> </tr> <tr> <td>Commanded position</td> <td>010001</td> <td>020001</td> <td>...</td> <td>040001</td> </tr> </tbody> </table> <p>Examples of setting for each purpose are shown below.</p> <ul style="list-style-type: none"> • Synchronized tapping setting: 3rd servo axis FB (000300) - 1st spindle FB (010000) • High-accuracy setting (roundness): 1st servo axis FB (000100) - 2nd servo axis FB (000200) • Spindle synchronization setting: 1st spindle FB (010000) - 2nd spindle FB (020000) 	Servo axis	1stAX	2ndAX	...	16thAX	Feedback position	000100	000200	...	000F00	Commanded position	000101	000201	...	000F01	Spindle	1stSP	2ndSP	...	4thSP	Feedback position	010000	020000	...	040000	Commanded position	010001	020001	...	040001	-
Servo axis	1stAX	2ndAX	...	16thAX																												
Feedback position	000100	000200	...	000F00																												
Commanded position	000101	000201	...	000F01																												
Spindle	1stSP	2ndSP	...	4thSP																												
Feedback position	010000	020000	...	040000																												
Commanded position	010001	020001	...	040001																												

Parameter	Details	Setting range (unit)												
Adr1 output unit : Adr8 output unit	Set output unit for the sampling data of the each address. <table border="1" data-bbox="485 342 1107 544"> <tr> <td>0</td> <td>No unit conversion</td> </tr> <tr> <td>B</td> <td>Outputs in micrometre</td> </tr> <tr> <td>C</td> <td>Outputs in sub-micrometre</td> </tr> <tr> <td>D</td> <td>Outputs in 10 nanometre</td> </tr> <tr> <td>E</td> <td>Outputs in 1 nanometre</td> </tr> <tr> <td>S</td> <td>Outputs in conventional interchangeable pulse of the spindle</td> </tr> </table> <p>(Note) For other than "0", set only data that internally complies with "#1004 ctrlunit (control unit)". However, this does not apply if the "address1 to 8" is set in the index No.method.</p>	0	No unit conversion	B	Outputs in micrometre	C	Outputs in sub-micrometre	D	Outputs in 10 nanometre	E	Outputs in 1 nanometre	S	Outputs in conventional interchangeable pulse of the spindle	0,B,C,D,E,S
0	No unit conversion													
B	Outputs in micrometre													
C	Outputs in sub-micrometre													
D	Outputs in 10 nanometre													
E	Outputs in 1 nanometre													
S	Outputs in conventional interchangeable pulse of the spindle													
Process Form	Set the sampling process type. <table border="1" data-bbox="469 725 1086 925"> <tr> <td>0</td> <td>One-shot The sampling stops when buffer is full.</td> </tr> <tr> <td>1</td> <td>Repeat valid After the process ends (the buffer is full), the "Trigger wait" state will be entered again.</td> </tr> <tr> <td>2</td> <td>Sampling buffer valid Sampling is executed as a ring buffer.</td> </tr> </table> <p>(Note) If a value other than "0" is set, the "End condition" must also be set. If not, the sampling process will not end until it is forcibly ended.</p>	0	One-shot The sampling stops when buffer is full.	1	Repeat valid After the process ends (the buffer is full), the "Trigger wait" state will be entered again.	2	Sampling buffer valid Sampling is executed as a ring buffer.	0 to 2						
0	One-shot The sampling stops when buffer is full.													
1	Repeat valid After the process ends (the buffer is full), the "Trigger wait" state will be entered again.													
2	Sampling buffer valid Sampling is executed as a ring buffer.													
End condition	Select the condition for ending the sampling process. (Note 4) <table border="1" data-bbox="505 1120 1134 1509"> <tr> <td>0</td> <td>Sampling completed When the buffer is full, the sampling ends.</td> </tr> <tr> <td>1</td> <td>Variable No. Sampling ends when the variable set in "variable No." is 0 or a value other than null (Note 2)</td> </tr> <tr> <td>2</td> <td>PLC device Sampling ends at the rising edge of the signal set in "PLC device". (Note 3)</td> </tr> <tr> <td>3</td> <td>Address condition, true Sampling ends when "address", "data" and "data mask" conditions are true. (Note 1)</td> </tr> <tr> <td>4</td> <td>Address condition, false Sampling ends when "address", "data" and "data mask" conditions are false. (Note 1)</td> </tr> </table>	0	Sampling completed When the buffer is full, the sampling ends.	1	Variable No. Sampling ends when the variable set in "variable No." is 0 or a value other than null (Note 2)	2	PLC device Sampling ends at the rising edge of the signal set in "PLC device". (Note 3)	3	Address condition, true Sampling ends when "address", "data" and "data mask" conditions are true. (Note 1)	4	Address condition, false Sampling ends when "address", "data" and "data mask" conditions are false. (Note 1)	0 to 4		
0	Sampling completed When the buffer is full, the sampling ends.													
1	Variable No. Sampling ends when the variable set in "variable No." is 0 or a value other than null (Note 2)													
2	PLC device Sampling ends at the rising edge of the signal set in "PLC device". (Note 3)													
3	Address condition, true Sampling ends when "address", "data" and "data mask" conditions are true. (Note 1)													
4	Address condition, false Sampling ends when "address", "data" and "data mask" conditions are false. (Note 1)													

Parameter	Details	Setting range (unit)				
Variable No.	<p>This sets the variable No. that functions as the start/end trigger. If a number that does not exist is input, an error will occur.</p> <table border="1"> <tr> <td>0</td> <td>System variable (#1299)</td> </tr> <tr> <td>Other than 0</td> <td>Designated common variable (#100 or more, #500 or more)</td> </tr> </table> <p>(Note) If both "Start condition" and "End condition" are set to other than "1" (variable No.), the value will be ignored even if set.</p>	0	System variable (#1299)	Other than 0	Designated common variable (#100 or more, #500 or more)	0 to 999
0	System variable (#1299)					
Other than 0	Designated common variable (#100 or more, #500 or more)					
PLC device	<p>This sets the PLC device that functions as the start/end trigger. When "*" is added at the head, the device will be a b contact. If a device that does not exist is input or the device No. is not within the setting range, an error will occur.</p> <table border="1"> <tr> <td>0</td> <td>Data sampling trigger signal (Y72C)</td> </tr> <tr> <td>Other than 0</td> <td>Designated device</td> </tr> </table> <p>(Note) If both "Start condition" and "End condition" are set to other than "2" (PLC device), the value will be ignored even if set.</p>	0	Data sampling trigger signal (Y72C)	Other than 0	Designated device	(*)X0000 to (*)X1FFF (*)Y0000 to (*)Y1FFF
0	Data sampling trigger signal (Y72C)					
Other than 0	Designated device					
Address	<p>This sets the address targeted for applying "Data mask". (Note 1) However, if an illegal address (highest-order bit 0) is input, an error will occur.</p> <p>(Note) If both "Start condition" and "End condition" are set to other than "3" or "4" (address conditions, true/false), the value will be ignored even if set. This item can be set with the same methods as "address 1 to 8".</p>	-				
Data	<p>This sets the data for judging the results of applying "Data mask" on "Address".</p> <p>(Note) If both "Start condition" and "End condition" are set to other than "3" or "4" (address conditions, true/false), the value will be ignored even if set.</p>	-				
Data mask	<p>This sets the data mask to be applied on "Address".</p> <p>(Note) If both "Start condition" and "End condition" are set to other than "3" or "4" (address conditions, true/false), the value will be ignored even if set.</p>	-				

(Note1) When "3" or "4" is set in "Start condition", be sure to set the address, the data, and the data mask. This applies to "End condition" as well. If these are not set, sampling may not start. When the same values are set in both "Start condition" and "End condition", sampling will not take a place.

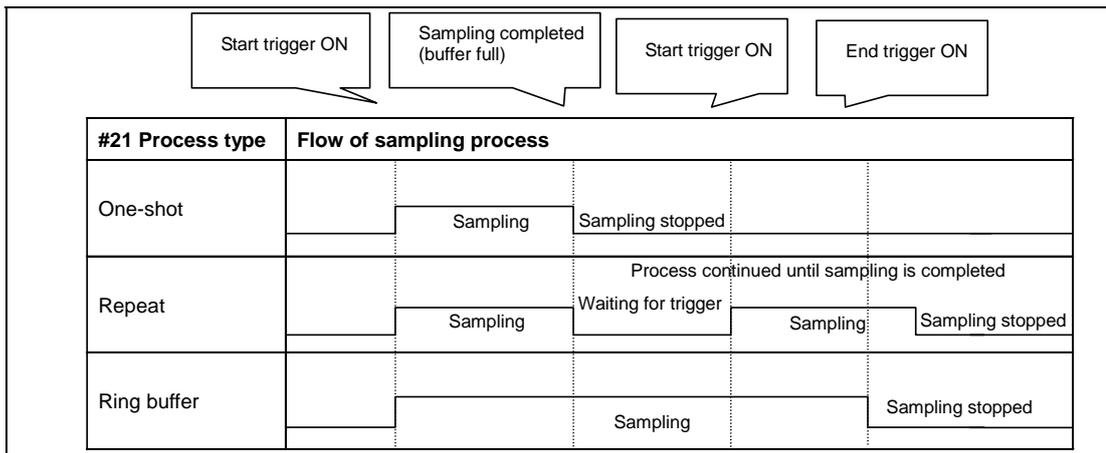
Example of "End condition" by setting address	
Setting	End condition : 3 Address : 12345678 Data : 0000100 Data mask : 0000FFFF
Meaning of setting	The trigger for sampling end turns ON when the results obtained by applying data mask "0000FFFF" on the data of which address is "12345678" matches "0000100".

(Note 2) When "1" is set in "Start condition"/"End condition", the common variable uses the floating decimal point type. Thus, if the calculation results are used, an error will occur, and the value may not be recognized as "0" because of its calculation error.

When using a multi-part system, the trigger will turn ON when the conditions are satisfied even if only one part system is used.

(Note 3) The variable data settings are validated immediately only from the program. These will not be validated immediately with settings from the screen or an external input.

(Note 4) Even if "End condition" is satisfied, the subsequent flow will differ according to the value set in "Process type". (This also applies when ending manually.)



5.8.1 Executing NC Data Sampling

Sampling is started when the menu is pressed.

Operation method (Executing NC data sampling)

(1)

(2)



The sampling starts and the menu cannot be selected. When the sampling end condition is satisfied, the state will be "Sampling stop". The menu can be selected.

(Note 1) The sampling will not end even if transferred to another screen in sampling.

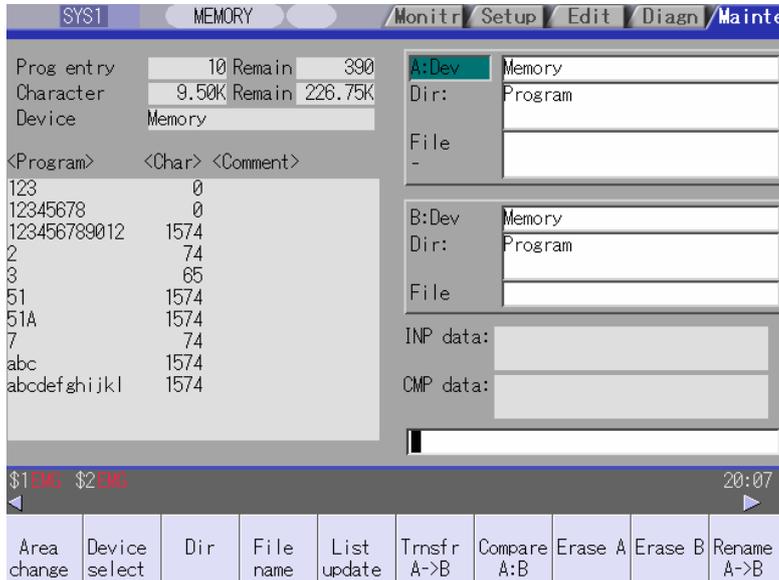
(Note 2) The data set in this function will not be output to the parameter.

(Note 3) The contents of "State" is returned to "Sampling stop". (The parameters related to NC data sampling is maintained.)

(Note 4) The data is displayed in 16 digits. However, the data can be set up to 8 digits.

5.8.2 Outputting Sampling Data

Sampling can be output on the data I/O screen under [Mainte] tag.



Operation method (Outputting the sampling data)

- (1) Select the data I/O screen under [Mainte] tag.
- (2) Set the following items in the file setting column A.

A:Dev:	Memory
Dir:	/LOG
File:	NCSAMP.CSV
- (3) Set the destination device, directory, file name in the setting column B.
- (4) Press the menu Trnsfr A->B. ➡ Sampling file will be output.

(Note 1) The data can not be output when the sampling is not executed.

(Note 2) The data can be output only when the sampling output valid parameter (#1224 aux08/bit0) is valid.

(Note 3) The output format differs depending on the sampling parameter "Output form".

(Note 4) The header information is not added to output sampling data.

Output file format (when output in decimal figure)

Sampling data is output in decimal figure when the sampling parameter "Output form" is set to "0".

(Example) When the number of channel is 3, the address is 1 to 3, and the output unit is E, it stops when 1ch is 100mm, 2ch is -100mm and 3ch is 50mm.

	Output format		
	1ch	2ch	3ch
1st data	200000000	-200000000	100000000
2nd data	200000000	-200000000	100000000
3rd data	200000000	-200000000	100000000
:	:	:	:

Output file format (when output in hexadecimal figure)

Sampling data is output in hexadecimal figure with 8 digits when the sampling parameter "Output form" is set to "1".

The output data is the sampling buffer dumped in the length of the long type data (32bits).

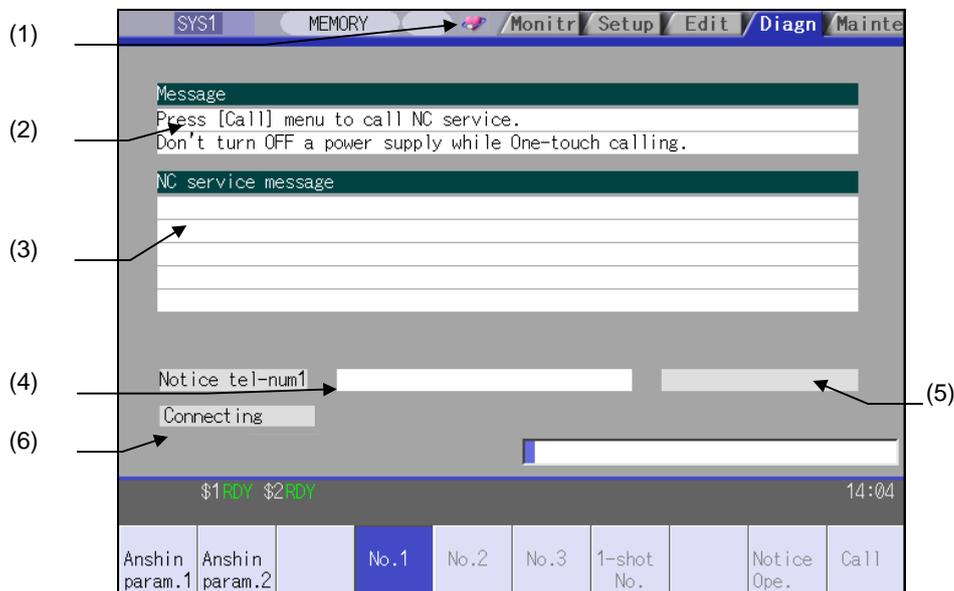
(Example) When the number of channel is 3, the address is 1 to 3, and the output unit is C, it stops when 1ch is 100mm, 2ch is -100mm and 3ch is 50mm.

Channel	Output format
1st data	1ch 0001E840
	2ch FFE17B80
	3ch 000F4240
2nd data	1ch 0001E840
	2ch FFE17B80
	3ch 000F4240
3rd data	1ch 0001E840
	2ch FFE17B80
	3ch 000F4240
:	:

5.9 Anshin-net Screen

5.9.1 Message Display Screen

Anshin-net connection status and message from NC service are displayed on this screen. Operator can make one-touch calls; validate operator notification; and change the notifying party's telephone No.



Display items

Display item	Details
(1) Anshin-net connection status	Flickers during Anshin-net communications. No displays during off-line state.
(2) Message	Message (Anshin-net connection status, etc.) is displayed.
(3) NC service message	Message from NC service is displayed. Up to 5 lines with 71 characters in each line can be displayed. (Note 1) If a message does not fit in the display area, the overflowed message is truncated. (Note 2) The message will be held until the server is connected again; the power is turned OFF; or operator notification operation is validated. (Note 3) When a message is received while Anshin-net screen is not displayed, the message will be displayed as soon as the screen is opened.
(4) Notice tel-num	Telephone No. of the notifying party is displayed.
(5) Comment	Comment concerning the notifying party is displayed.
(6) Communication status message	Message on communication status is displayed. "Transmission completed" and "Reception completed" displays will go out at screen transitions.

Menus

Menu	Details	Type	Reference
Anshin param.1	Jumps to the Anshin-net parameter1 screen.	C	"Setting the parameter"
Anshin param.2	Jumps to the Anshin-net parameter 2 screen. (Note) This menu can be used only when a password has been set by the machine tool builder.	C	"Setting the parameter"
No.1	This changes the notifying party's telephone No. to the number at Notice tel-num1 in the Anshin-net parameter 1 screen.	B	
No. 2	This changes the notifying party's telephone No. to the number at Notice tel-num2 in the Anshin-net parameter 1 screen.	B	
No. 3	This changes the notifying party's telephone No. to the number at Notice tel-num3 in the Anshin-net parameter 1 screen.	B	
1-shot No.	This changes the notifying party's telephone No. to the number set arbitrarily.	A	"Setting the arbitrary telephone No."
Notice Ope.	This validates/invalidates the operator notification.	B	"Making the operator notification valid"
Call	This makes a one-touch call.	B	"Making a one-touch call"

5.9.2 Anshin-net Parameter 1, 2 Screen

The notifying party's telephone No. and comments are set in this screen.

No.	Name	Data
10801	Notice tel num 1	
10802	Comment 1	
10803	Notice tel num 2	
10804	Comment 2	
10805	Notice tel num 3	
10806	Comment 3	
10807	Password	
10808	User number	
10812	Anet enable	0

Anshin-net parameter 1

#	Item	Details	Setting range (unit)
10801	Notice tel num 1	Set the call-back telephone No. used for one-touch call and operator notification. Begin with the No. from an area code for domestic call. Begin with a communication company No. for international call. Hyphens "-" can be used as a delimiting character.	Within 28 characters
10802	Comment 1	Set a comment, such as a party's name, for the notification party telephone No.1.	Within 20 alphanumerical characters (excluding spaces)
10803	Notice tel num 2	Set the call-back telephone No. used for one-touch call and operator notification. Begin with the No. from an area code for domestic call. Begin with a communication company No. for international call. Hyphens "-" can be used as a delimiting character.	Within 28 characters
10804	Comment 2	Set a comment, such as a party's name, for the notification party telephone No.2.	Within 20 alphanumerical characters (excluding spaces)
10805	Notice tel num 3	Set the call-back telephone No. used for one-touch call and operator notification. Begin with the No. from an area code for domestic call. Begin with a communication company No. for international call. Hyphens "-" can be used as a delimiting character.	Within 28 characters
10806	Comment 3	Set a comment, such as a party's name, for the notification party telephone No.3.	Within 20 alphanumerical characters (excluding spaces)
10807	Password	Set the password for sharing of machining data.	4 characters (one-byte alphanumeric characters, without space)
10808	Customer number	Set the user No. for sharing of machining data.	Within 8 characters (one-byte alphanumeric characters, without space)
10812	Anshin-net valid	Select whether to enable the Anshin-net function.	0: Disable 1: Enable

5.9.3 Operation Method

Operation method (Executing NC data sampling)

- (1) Set sampling condition and data. ➔ Notice Ope. menu key is highlighted and the other menus will be disabled.
"Operator notice effective" is displayed.

When the Notice Ope. menu key is pressed again while operator notification is valid, menu's highlight and disabled state will be canceled, and the operator notification will be disabled.

If machining ends normally/abnormally while operator notification is valid, system is connected to the NC service.

("Dialing" → "Waiting for dialing" → "Verifying the user registration" → "Connecting" → "Sending" → "Connecting" → "Transmission completed")

The message from NC service is displayed.

After data has been sent, menu's highlight and disabled state is canceled, and operator notification will be disabled.

A call is made to the currently displayed notifying party's telephone No. from the call center.

Making a one-touch call

- (1) Press the Call menu key. ➔ Call menu is highlighted and the other menus will be disabled.
"Carry out one-touch call? (Y/N)" is displayed.
- (2) Press "Y" or the INPUT key. ➔ Connects to NC service. Connection status is displayed in order.
("Dialing" → "Waiting for dialing" → "Verifying the user registration" → "Connecting" → "Sending" → "Connecting" → "Transmission completed")
Diagnosis data is sent to NC service after connection has been established.
After diagnosis data has been sent, menu's highlight and disabled state is canceled, and the message from NC service is displayed.

Setting the arbitrary telephone No.

- (1) Press the 1-shot No. menu key. ➔ 1-shot No. menu is highlighted and the other menu will be disabled.
The mode for setting the arbitrary No. is entered.
The cursor is displayed at the notifying party's telephone No. column.
- (2) Enter numerical value.
012-345-6789 INPUT ➔ Menu's highlight and disabled state are canceled, and "012-345-6789" is displayed at the cursor position.
The notifying party's telephone No. is changed to the number entered.

If the 1-shot No. menu key is pressed again after an arbitrary telephone No. has been set, the notifying party's telephone No. will be the number already selected (No. 1, 2, or 3).

(Note 1) If the No.1, No.2, or No.3 menu key is pressed after an arbitrary telephone No. has been input, the displayed arbitrary telephone No. will be cleared and the notifying party's telephone No. will be changed to the number 1, 2, or 3 on the Anshin-net parameter 1 screen.

Setting the parameter

- (1) Press the menu key in the message display screen.  Anshin-net parameter 1 screen is displayed. (Press the menu key to set the parameter for Anshin-net parameter 2 screen.)
- (2) Move the cursor to the item to be set with the and keys.  The cursor is moved to the item to be set, and the value currently set in the data setting area is displayed.
- (3) Enter a value and press the key.  The input value is reflected at the cursor position.

Press the cancel key to return to the message display screen.

5.9.4 Sharing Machining Data

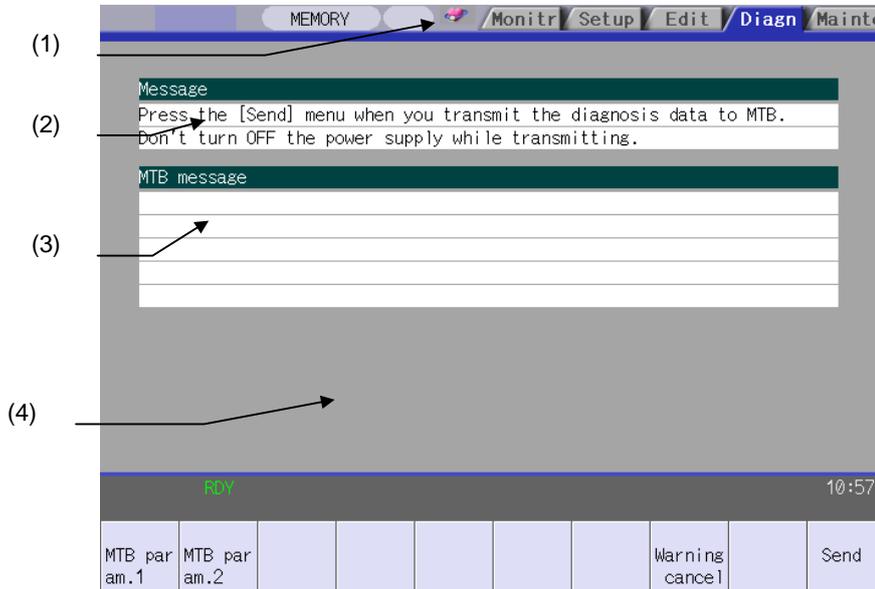
Machining data can be send to/received from Anshin-net server in the call center.

The transmission and reception of data can be operated on the Edit screen and the Maintenance screen. Refer to "4.5.14 Sharing Machining Data" and "6.2.13 Sharing Machining Data" for the operation method.

5.10 Machine Tool Builder Network System (MTB net) Screen

5.10.1 Message Display Screen

Diagnosis data can be transmitted to the machine tool builder from this screen. The net connection state and messages from the machine tool builder are also displayed on this screen.



Display items

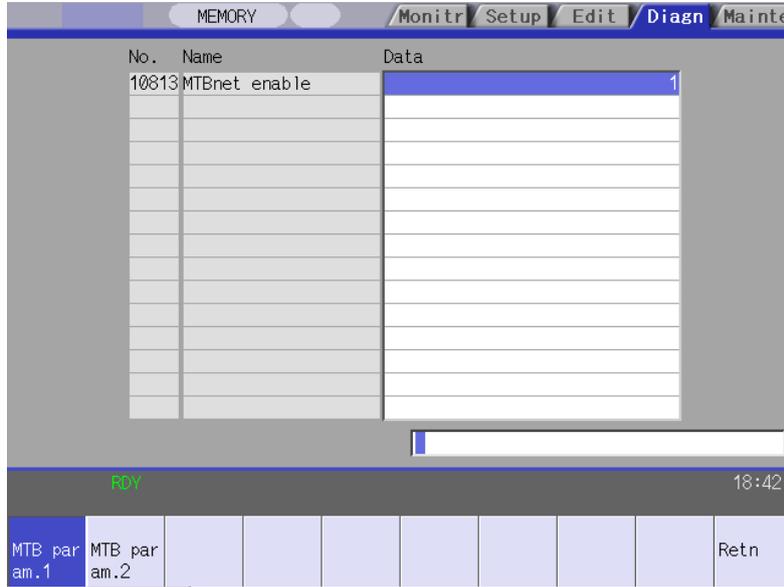
Display item	Details
(1) Machine tool builder network connection status	Flickers during machine tool builder network communications. No displays during off-line state.
(2) Message	Message (machine tool builder network connection status, etc.) is displayed.
(3) MTB message	Message from machine tool builder is displayed. Up to 5 lines with 71 characters in each line can be displayed. (Note 1) If a message does not fit in the display area, the overflowed message is truncated. (Note 2) The message will be held until the power is turned OFF; or next message from machine tool builder is received. (Note 3) When a message is received while machine tool builder network screen is not displayed, the message will be displayed as soon as the screen is opened.
(4) Communication status message	Message on communication status is displayed. "Transmission completed" and "Reception completed" displays will go out at screen transitions.

Menus

Menu	Details	Type	Reference
MTB param.1	Jumps to the MTB net parameter1 screen. (Note 1) This menu cannot be selected while connecting the machine tool builder network.	B	"Setting the parameter"
MTB param.2	Jumps to the MTB net parameter2 screen. (Note 1) This menu can be used only when a password has been set by the machine tool builder. (Note 2) This menu cannot be selected while connecting the machine tool builder network.	B	"Setting the parameter"
Warning cancel	This cancels the warning of network service.	C	"Canceling a warning of network service"
Send	This transmits diagnosis information to the machine tool builder.	A	"Transmitting diagnosis information to the machine tool builder"

5.10.2 MTB Net Parameter 1, 2 Screen

The valid/invalid of the machine tool builder network (MTB net) function is set in this screen.



Menus

Menu	Details	Type	Reference
MTB param.1	Jumps to the MTB net parameter1 screen.	B	"Setting the parameter"
MTB param.2	Jumps to the MTB net parameter2 screen. (Note 1) This menu can be used only when a password has been set by the machine tool builder.	B	"Setting the parameter"
Return	Jumps to the MTB net screen.	C	"Canceling a warning of network service"

MTB net parameter 1

#	Item	Details	Setting range (unit)
10813	MTBnet enable	Select whether to enable the machine tool builder network system.	0: Disable 1: Enable Standard setting: 0 (Note) Values other than "0" and "1" are invalid.

5.10.3 Operation Method

Transmitting diagnosis information to the machine tool builder

- (1) Press the menu key **Send**. → The **Send** menu will be highlighted, and the message "Transmit diagnosis data? (Y/N)" will appear.
- (2) Press "Y" or the **INPUT** key. → Connects to the machine tool builder. Connection status is displayed in order. Diagnosis data is sent to the machine tool builder after connection has been established. After diagnosis data has been sent, menu's highlight is canceled, and the message from the machine tool builder is displayed.

Canceling a warning of network service

- (1) Press the menu key **Warning cancel**. → The warning message from network service is cleared.

(Note 1) A warning message will appear if an error occurs when reception of the diagnosis results is completed or during communication.

(Note 2) NC reset can also clear the warning message from network service.

Setting the parameter

- (1) Press the **MTB param.1** menu key in the message display screen. → MTB net parameter 1 screen is displayed. (Press the **MTB param.2** menu key to set the parameter for MTB net parameter 2 screen.)
- (2) Move the cursor to the item to be set with the **↑** and **↓** keys. → The cursor is moved to the item to be set, and the value currently set in the data setting area is displayed.
- (3) Enter a value and press the **INPUT** key. → The input value is reflected at the cursor position.

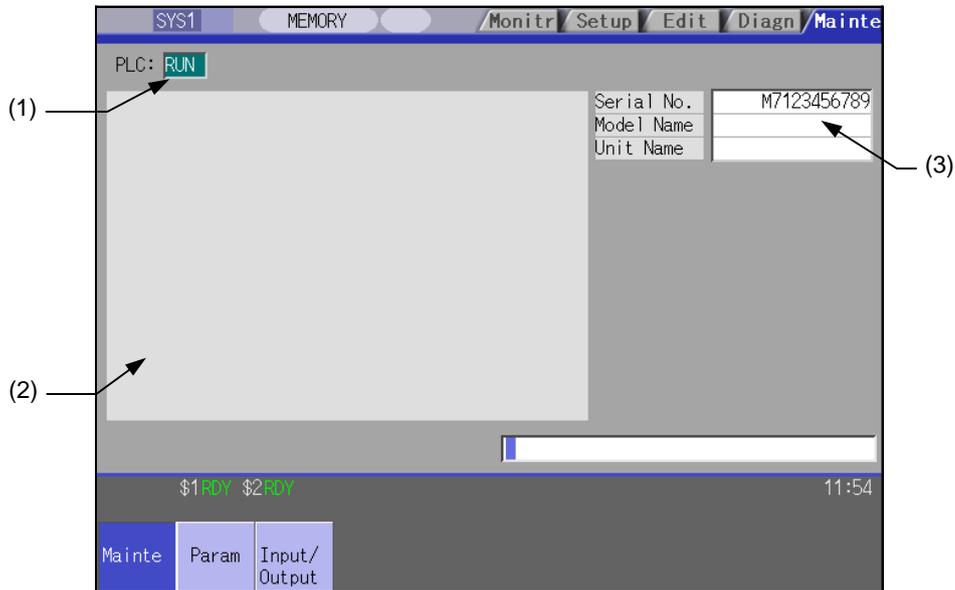
(Note 1) To return to the message display screen, **Return** key or **CANCEL** key is pressed.

(Note 2) When the communication between the call center and the machine tool builder is started while displaying MTB net parameter screen 1 or 2, the parameters cannot be changed until the communication is completed. The operation message "Network service is connected" appears.

6. Maintenance Screens

6. Maintenance Screens

The NC memory can be formatted, the absolute position parameters setting, the SRAM backup, etc. are possible on this screen. Important operations are protected with a password.



Display items

Display item	Details
(1) PLC stop state	The PLC state (stopped/stop canceled) is displayed. Stopped : "STOP" is highlighted. Stop canceled : "STOP" is not highlighted.
(2) Explanation of menu operations	A brief explanation of the menu operations is displayed.
(3) Serial No.	Currently set NC information (serial No., model name and unit name) is displayed.

6. Maintenance Screens

Menus

Menu	Details	Type	Reference
Psswd input	This changes the screen related to the maintenance by setting the password.	A	
PLC stop	This forcible stops the PLC ladder process. If this menu key is pressed in the stopped state, the stop is canceled.	C	
All backup	This backs up (saves) or restores (reloads) the file such as SRAM etc. to designated device.	C	
System setup	This automatically executes necessary parameter setting for driving servomotor only by setting necessary minimum item.	C	
Adjust S-ana	This changes the screen to that for adjusting the spindle analog output.	C	
To Abs pos	This sets the absolute position for servo axis (arbitrary NC axis, PLC axis).	C	
AUX test	This carries out absolute position setting or test operation by the auxiliary axis forward run/reverse run. This menu is only for 700 series.	C	
Collect set	This executes the followings. Setting to collect the diagnosis data Confirming the diagnosis data collection status Starting/Stopping to collect the diagnosis data Clearing the diagnosis data	C	
Format	This formats the NC memory.	B	
T-life format	This formats the tool life management data.	B	
Serial No.Set	This changes the NC serial No.	A	
Console exec	The console is executed. The MS-DOS window will appear.	C	
To In/out	This changes the screen to the Data Input/Output screen.	C	
To param	This changes the screen to the Parameter screen.	C	
SRAM backup	<700 series> This backs up the NC SRAM information on the HD. <70 series> This backs up the NC SRAM information on the memory card.	B	
HMI Quit	This quits the screen operation.	B	

Backing up the NC SRAM information

- (1) Press the menu . ➔ A message appears to confirm the backup execution.
- (2) Press or . ➔ The backup is executed.
 Press a key other than or to cancel the backup.
 - <700 series>
The data is backed up into "D:\ NCFILE \ SRAM.BIN" on the HD.
(If SRAM.BIN already exists, the original file will be saved as SRAM.BAK.)
 - <70 series>
The data is backed up into the memory card.

6.1 Parameter Screens

The various parameters can be displayed and set on this screen.
 The configuration of the all parameter screens is applied to one of the following five patterns.

<Pattern 1> This is the screen for setting the common parameters for the axis, device, etc.
 (Process param, etc.)

No.	Name	Data	No.	Name	Data
1026	Base axis I	X			
1027	Base axis J	Y	8001	<WRK COUNT>	
1028	Base axis K	Z	8001	WRK COUNT M	0
1029	Flat axis I	X	8002	WRK COUNT	0
1030	Flat axis J	Y	8003	WRK COUNT LIMIT	0
1031	Flat axis K	Z		<AUTO TLM>	
1084	Arc error	0.100	8004	SPEED	0
1171	Tap return ovr	0	8005	ZONE r	0.000
1185	Feedrate F1	500	8006	ZONE d	0.000
1186	Feedrate F2	500		<AUTO CORNER OVR.>	
1187	Feedrate F3	500	8007	OVERRIDE	50
1188	Feedrate F4	500	8008	MAX ANGLE	120
1189	Feedrate F5	500	8009	DSC_ZONE	20.000
1506	F1 upper limit	10000		<T-TIP OFFSET>	
1507	F1 change constant	10	8010	ABS.MAX	0.000
			8011	INC.MAX	0.000

\$1 000 \$2 000 14:11

Process param Ctrl param 1 Ctrl param 2 Axis param Barrier data Param number Area copy Area paste

<Pattern 2> This is the screen for setting the parameters having an array structure for each axis
 (Axis param, etc.)

No.	Name	X	YM	YS	Z
1063	Manual dog-type	0	0	0	0
8201	AX. RELEASE	0	0	0	0
8202	OT-CHECK OFF	1	1	1	1
8203	OT-CHECK-CANCEL	0	0	0	0
8204	OT-CHECK-N	-20.000	-20.000	-461.000	-121.000
8205	OT-CHECK-P	20.000	20.000	1.000	31.000
8206	TOOL CHG.P	0.000	0.000	0.000	0.000
8207	G76/87 IGNR	0	0	0	0
8208	G76/87 (-)	0	0	0	0
8209	G60 SHIFT	0.000	0.000	0.000	0.000
8210	OT INSIDE	0	0	0	0
8211	MIRR. IMAGE	0	0	0	0
8212					
8213	Rotation axis type	0	0	0	3

\$1 000 \$2 000 14:12

Process param Ctrl param 1 Ctrl param 2 Axis param Barrier data Param number Area copy Area paste Next axis

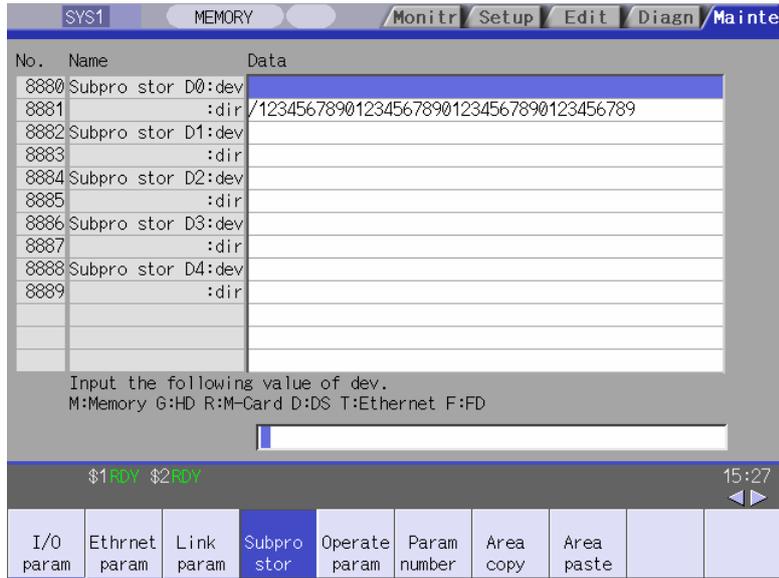
<Pattern 3> This is the screen for setting parameters common for the axis and device, etc. The parameter names are not displayed (Machine error data, PLC constant, etc.)

No.	Data	No.	Data	No.	Data	No.	Data
18001	0	18016	0	18031	0	18046	0
18002	0	18017	0	18032	0	18047	0
18003	0	18018	0	18033	0	18048	0
18004	0	18019	0	18034	0	18049	0
18005	0	18020	0	18035	0	18050	0
18006	0	18021	0	18036	0	18051	0
18007	0	18022	0	18037	0	18052	0
18008	0	18023	0	18038	0	18053	0
18009	0	18024	0	18039	0	18054	0
18010	0	18025	0	18040	0	18055	0
18011	0	18026	0	18041	0	18056	0
18012	0	18027	0	18042	0	18057	0
18013	0	18028	0	18043	0	18058	0
18014	0	18029	0	18044	0	18059	0
18015	0	18030	0	18045	0	18060	0

<Pattern 4> This is the screen for setting parameters having an array structure for each part system (Base system parameters, etc.)

No.	Name	\$1	\$2	\$3	\$4
1001	SYS_ON	1	1	0	0
1002	axisno	4	3	0	0
1003	iunit	E	E	D	E
1004	ctrl_unit	E	E	E	E
1005	plcunit	B	B	D	E
1006	mcpunit	B	B	D	E
1025	I_plane	1	1		
1026	base_I	X	X		
1027	base_J	Y	Y		
1028	base_K	Z	Z		
1029	aux_I				
1030	aux_J				
1031	aux_K				

<Pattern 5> This is the screen for setting the subprogram storage destination parameters



Menus

Menu	Details	Type	Reference
Param No.	A arbitrary parameter No. can be selected. When the parameter No. is set and the INPUT key is pressed, the parameters will appear with that No. at the head. The cursor will also move to that No.	A	-
Area copy	This copies the parameter setting values in the designated range. The range is designated with numbers.	A	6.1.4 Copying/Pasting Parameters
Area paste	This pastes the range of parameters designated in area copy. They are pasted in a parameter corresponding to the axis or part system where the cursor is. Once copied, a parameter can be pasted any number of times until a new parameter is copied.	B	
Next axis	This can be selected when there are five or more display axes in the selected part system. This is used in the screen for the parameters having an array structure for each axis.	C	-
Next system	This can be selected when there are two or more screen display part systems. Use this with the parameter screen for each parameter.	C	-
	This can always be selected when the base part system parameter screen is open. The display changes to the PLC axis display when this menu is pressed.		-
Process param	This changes the screen to the user parameter screen. (Note) The barrier data is displayed only for the L specifications.	C	6.1.5 User Parameters
Ctrl param 1			
Ctrl param 2			
Axis param			
Barrier data			
I/O param			
Ethernet param			
Link param			
Subpro stor			
Operate param			

Menu	Details	Type	Reference
BaseSys param	<p>This changes the screen to the Machine parameter screen.</p> <p>(Note 1) Normally, the machine parameters can be referred to, but cannot be set.</p> <p>(Note 2) The "RotAxis param", "AUX param", "Open param 1" and "Open param 2" menus are displayed only when these options are valid.</p> <p>(Note 3) The "AUX param" is displayed for 700 series only.</p>	C	-
BaseAx spec			
BaseCom param			
Axis spec			
Zp-rtn param			
Abs pos param			
Servo param			
Spindle spec			
Spindle param			
RotAxis param			
PLC timer			
PLC inc timer			
PLC counter			
PLC constnt			
Bit select			
Er Comp param			
Er Comp data			
Macro list			
Posn switch			
AUX param			
Open param1			
Open param2			
CC-Link param1			
CC-Link param2			
PLCIndx param			

6.1.1 Changing the Parameter Display

Changing to the computer link parameters

(1) Press the menu change key until the menu **Link param** appears.

The menu **Link param** appears.

(2) Press the menu **Link param**.

The computer link parameters appears.
The menu **Link param** is highlighted.

No.	Name	Data	No.	Name	Data
9601	BAUD RATE	0	9616	CTRL. INTERVAL	0
9602	STOP BIT	0	9617	WAIT TIME	0
9603	PARITY EFFECTIVE	0	9618	PACKET LENGTH	0
9604	EVEN PARITY	0	9619	BUFFER SIZE	0
9605	CHR. LENGTH	0	9620	START SIZE	0
9606	HAND SHAKE	0	9621	DC1 OUT SIZE	0
9607	TIME-OUT SET	0	9622	POLLING TIMER	0
9608	DATA CODE	0	9623	TRANS. WAIT TMR	0
9609	LINK PARAM. 1	00	9624	RETRY COUNTER	0
9610	LINK PARAM. 2	00	9625		
9611	LINK PARAM. 3	00	9626		
9612	LINK PARAM. 4	00	9627		
9613	LINK PARAM. 5	00	9628		
9614	START CODE	0	9629		
9615	CTRL. CODE OUT	00	9630		

\$1000 \$2000 20:41

I/O param	Ethnet param	Link param	Subpro stor		Area copy	Area paste	
-----------	--------------	-------------------	-------------	--	-----------	------------	--

6.1.2 Setting the Parameters

The method of setting the parameters is explained in this section. For the setting range of each parameter, refer to "3.8.8 Parameter list".

Press the menu key to select the parameter screen, and select the data to be set with the cursor key.

"8201 Axis removal" Setting Y1 axis to "1"

(1) Press the menu .

(2) Use the , , , and keys to move the cursor to the position to be set.

The cursor can also be moved with the menu .
Refer to "6.2.2 Selecting the parameter No.".



The cursor moves to the position of the setting target data.

No.	Name	X1	Y1
8201	AX. RELEASE	0	0
8202	OT-CHECK OFF	1	1
8203	OT-CHECK-CANCEL	0	0
8204	OT-	1.000	1.000
8205	OT+	1.000	1.000
8206	TOOL CHG.P	0.000	0.000

The current setting value is displayed in the input area.

(3) Input the value.
1



The setting value appears, and the cursor moves.

No.	Name	X1	Y1
8201	AX. RELEASE	0	1
8202	OT-CHECK OFF	1	1
8203	OT-CHECK-CANCEL	0	0
8204	OT-	1.000	1.000
8205	OT+	1.000	1.000
8206	TOOL CHG.P	0.000	0.000

"8205 OT+" Setting X1 axis to "100.0" and Z1 axis to "200.0"

(1) Press the menu .

(2) Use the and keys to move the cursor to the position to be set. (The row does not need to be designated with the and keys.)



The cursor moves to the position of the setting target data.

No.	Name	X1	Y1	Z1
8201	AX. RELEASE	0	1	0
8202	OT-CHECK OFF	1	1	1
8203	OT-CHECK-CANCEL	0	0	0
8204	OT-	1.000	1.000	1.000
8205	OT+	1.000	1.000	1.000
8206	TOOL CHG.P	0.000	0.000	0.000

The current setting value is displayed in the input area.

(3) Input the value.
(Format: 1st row/2nd row/3rd row/4th row)

100//200



The setting value appears, and the cursor moves.

No.	Name	X1	Y1	Z1
8201	AX. RELEASE	0	1	0
8202	OT-CHECK OFF	1	1	1
8203	OT-CHECK-CANCEL	0	0	0
8204	OT-	1.000	1.000	1.000
8205	OT+	100.000	1.000	1.000
8206	TOOL CHG.P	0.000	0.000	0.000

(Note 1) If appears at the lower right of the screen when the parameter value is changed, the parameter value will be validated when the power is turned ON again.

(Note 2) If the key is pressed without inputting a value, the cursor will move without changing the parameter setting value.

(Note 3) If a character string, such as an axis name or input/output device name, is set in the parameter, the setting will be cleared when 0 is input and the key is pressed.

(Note 4) Parameters for up to four rows, which are as currently displayed, can be set at once.

(Note 5) If the parameter value for several rows is input simultaneously, the values will be set from the currently displayed left end no matter which row the cursor is currently at.

6.1.3 Copying/Pasting Parameters

Designating the cursor position and copying

- (1) Move the cursor to the parameter to be copied.
- (2) Press the menu and .



The menu is highlighted.
The setting value of the parameter at the cursor position highlighted.

No.	Name	X1
8201	AX. RELEASE	0
8202	OT-CHECK OFF	1
8203	OT-CHECK-CANCEL	0
8204	OT-	1.000
8205	OT+	100.000
8206	TOOL CHG.P	0.000

(Note) When the screen has an array structure for each axis or each part system, an error will occur if different rows (axis or part system) are designated for the copy start position and end position.

Copying by designating the parameter No. with key inputs

(1) Move the cursor to the display area of the axes or part system to be copied.

(2) Press the menu .

The menu is highlighted.

(3) Designate the copy range.
Format: First No./Last No.
8203/8205

The copy target range is highlighted.
The menu highlight returns to normal.

If the last No. to be copied is the last No. of the currently displayed parameter type, "E" can be designated. **(Example)** 8201/E
If only one parameter is being copied, input only the No. of the parameter to copy, and then press the key.

No.	Name	X1
8201	AX. RELEASE	0
8202	OT-CHECK OFF	1
8203	OT-CHECK-CANCEL	0
8204	OT-	1.000
8205	OT+	100.000
8206	TOOL CHG.P	0.000

(Note 1) The copy target range highlight returns to the normal display by pasting. The copy target range is valid until the parameter type display is changed.

(Note 2) Only the currently displayed parameter types can be copied.

(Example) If the parameter numbers #1001 to #1028 of the base system parameter are designated as the copy target range, the base axis specification parameters #1013 to #1024 cannot be copied.

Pasting the copied data

(1) Move the cursor to the display area of the axes or part system where the data will be pasted.

(2) Press the menu key .

The menu is highlighted, and a message confirming the operation appears.

(3) Press or .

The copied data is written to the parameter with the same number as when copied, in the area where the cursor is.
The menu highlight returns to normal.

The copied data will not be written in when the key is pressed.

(Note) If the corresponding parameter is changed after the copy target range is designated, the changed value will be pasted.

6.1.4 User Parameters

The user parameter configuration is as shown below.

Parameter type	Next axis menu	Next system menu	Area copy Area paste menu
Process Parameter	-	△ (Note 3)	△ (Note 3)
Control Parameter 1	-	-	-
Control Parameter 2	-	-	-
Axis Parameter	△ (Note 1)	-	△ (Note 2)
Barrier data ^(Note 5)	-	△ (Note 3)	○
Input/Output Parameter	-	-	-
Ethernet Parameter	-	-	-
Computer Link Parameter	-	-	-
Sub-program storage destination parameter	-	△ (Note 3)	△ (Note 3)
Operation Parameter	-	-	-

(Note 1) This is valid only when the total number of valid NC axes and PLC axes in the entire part system is five or more.

(Note 2) This is valid only when the total number of valid NC axes and PLC axes in the entire part system is two or more.

(Note 3) This is valid only when the number of valid part systems is two or more.

(Note 4) The barrier data appears only for the L specifications.

Operation methods (Subprogram storage destination parameters) Example 1

Setting the device (Example: Assign FD to D1)

(1) Set the cursor to the "#8882 Subpro stor
D1: dev" (D1 device).

(2) Input



"FD" appears at "#8882 D1: dev".
The cursor moves down one line.

(Note) Input the following values for the device name.

M: Memory, G: HD, R: Memory card, D: DS (Data server), F: FD
(In 70 series, only "Memory" and "Memory card" can be used.)

Operation methods (Sub program storage destination parameter) Example 2

Setting the directory (Example: Setting "/TEMP1/TEMP12/TEMP123" to D1)

(1) Set the cursor to "#8883 dir" (D1 directory)



The current "#8883 D1: dir" contents are echoed back to the input area.

(2) Set "/TEMP1/TEMP12/TEMP123", and press the



The value is set, and the cursor moves down one line.

6.1.5 Echo Back

If more than 14 characters can be set for the parameter, or for bit-related parameters, the parameter setting value at the cursor position appears at the input area. If more than 14 characters are set in the parameter, "... will appear in the data field.

No.	Name	Data	No.	Name	Data
9701	IP addr auto set	0	9719	Host1 Wrđ pos:cmnt	0
9702	IP address	0.0.0.0	9720	Host1 Wrđ num:cmnt	0
9703	Subnet mask	0.0.0.0	9721	Host1 no total siz	0
9704	Gateway	0.0.0.0			
9705	Timeout	0	9731	Host2 host name	
9706	Host No.	0	9732	Host2 user name	
			9733	Host2 password	*****
9711	Host1 host name		9734	Host2 directory	/bnc59/user/program
9712	Host1 user name		9735	Host2 host type	
9713	Host1 password	*****	9736	Host2 Wrđ pos:name	0
9714	Host1 directory		9737	Host2 Wrđ pos:size	0
9715	Host1 host type	0	9738	Host2 Wrđ pos:Dir	0
9716	Host1 Wrđ pos:name	0	9739	Host2 Wrđ pos:cmnt	0
9717	Host1 Wrđ pos:size	0	9740	Host2 Wrđ num:cmnt	0
9718	Host1 Wrđ pos:Dir	0	9741	Host2 no total siz	0

Parameters for which 14 or more characters can be set

No.	Data	No.	Data	No.	Data	No.	Data
6401	00000000	6416	00000000	6431	00000000	6446	00000000
6402	00000000	6417	00000000	6432	00000000	6447	00000000
6403	00000000	6418	00000000	6433	00000000	6448	00000000
6404	00000000	6419	00000000	6434	00000000	6449	10000011
6405	00000000	6420	00000000	6435	00000000	6450	00000100
6406	00000000	6421	00000000	6436	00000000	6451	00110000
6407	00000000	6422	00000000	6437	00000000	6452	00000000
6408	00000000	6423	00000000	6438	00000000	6453	00000001
6409	00000000	6424	00000000	6439	00000000	6454	00000000
6410	00000000	6425	00000000	6440	00000000	6455	00000000
6411	00000000	6426	00000000	6441	00000000	6456	00000000
6412	00000000	6427	00000000	6442	00000000	6457	00000000
6413	00000000	6428	00000000	6443	00000000	6458	00000000
6414	00000000	6429	00000000	6444	00000000	6459	00000000
6415	00100000	6430	00000000	6445	00000000	6460	00000000

Bit-related parameters

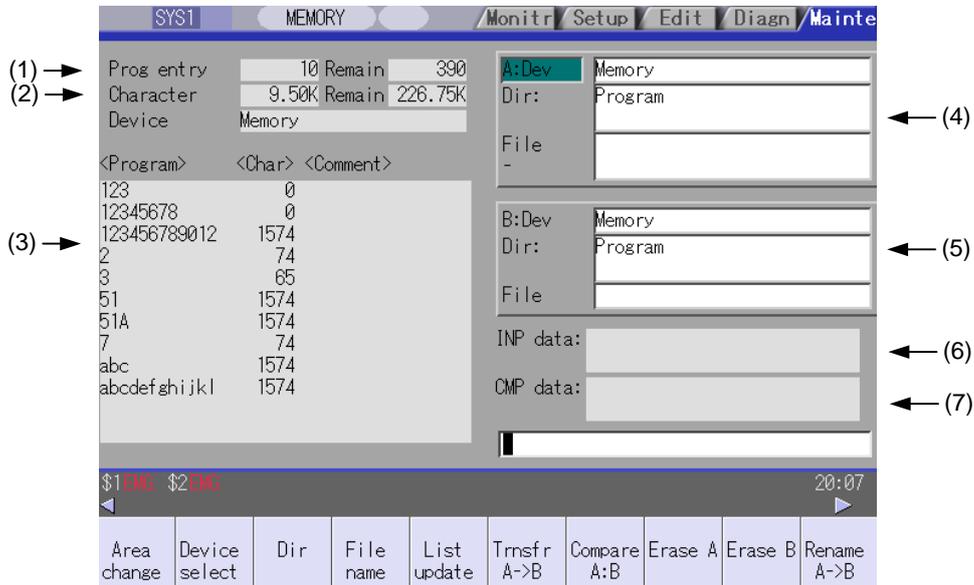
6.2 Input/Output Screen

The Input/Output screen is used to carry out NC data input/output between the NC internal memory and the external input/output devices.

Here, the hard disk built into the NC device is also treated as external devices.

In 70 series, only "Memory", "Memory card", "Serial", "Ethernet" and "Anshin-net server" can be used.

When the multi-part system program management is valid (#1285 ext21/bit0 =1, ext21/bit1 =1), the program input/output is executed across all part systems in batch if the selected device is NC memory.



Display items

Display item	Details
(1) Number of programs registered and remainder (Note 1)	This displays the registration information of machining program of the selected device. Number of programs registered : This displays the number of programs previously registered as user machining programs. Remainder : This displays the remaining number of programs that can be registered. When "Memory" is selected as the device, the total of the number of programs registered and the remainder is the maximum number of registrations set in the specifications.
(2) Number of memory characters and remainder (Note 1)	This displays the number of characters of the machining program of the selected device. Number of memory characters : This displays the number of characters previously registered as user machining programs. Remainder : This displays the remaining number of characters that can be registered. The total of the number of memory characters and the remainder is the maximum number of memory characters set in the specifications.

Display items	Details
(3) List (Note 2)	<p>This displays a contents list (directory and file name) of the directory in the setting column (file setting column A or B) where the cursor is currently located.</p> <p>Program : When "Memory" is selected for the device, this displays the file name (program No.) of the machining programs already registered. The file names are displayed in order from the smallest number, from 1 to 99999999. When a device other than memory is selected, this displays the file name and directory to be included in the directory that is set in the current setting column. When the number of characters exceeds 12, the excess is indicated as "***".</p> <p>Character : The size of each file (when memory is selected for the device, the number of characters in the machining program). When directory is selected, this displays "DIR".</p> <p>Comment : This displays the comment (up to 17 alphanumeric characters and symbols) of each file. The date which the file is updated is displayed for the HD, FD, memory card, DS or Ethernet. When the number of characters exceeds 17, the excess is not displayed.</p>
(4) File setting column A	This sets the device, directory, and file name of the target file for transfer, compare, erasing, etc., operations.
(5) File setting column B	When transferring, the file name of the transfer origin file is set. When renaming, the file name before renaming is set. When erasing, the erasing range is set. When the number of characters exceeds 28, the excess is not displayed.
(6) Input data	This displays the data being transferred.
(7) Comparison data	This displays the data being compared. If an error occurs during comparison, the block with the error is displayed.

(Note 1) Depending on the device, some items are not displayed.

Display item	Device							
	Memory	HD	Serial	Memory card	DS	Ethernet	FD	Anshin-net server
Number of programs registered	○	○	×	○	○	○	○	×
Remainder	○	×	×	×	×	×	×	×
Number of memory characters	○	○	×	○	○	○*	○	×
Remainder	○	○	×	○	○	×	○	×
List	○	○	×	○	○	○	○	×

○ : Displayed × : Not displayed

* : When the Ethernet parameter "#97*1 Host n no total siz" is set to 1, the number of host memory characters will not appear.

(Note 2) The list does not appear when using serial.

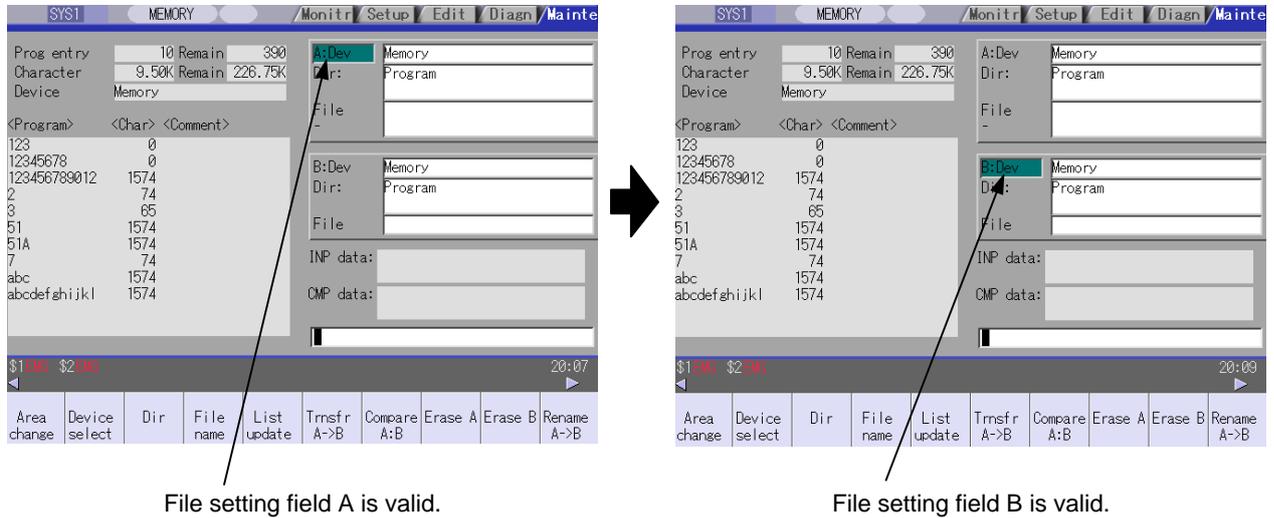
Menus

Menu	Details	Type	Reference
Area change	This changes the setting area to file setting column A (transfer origin) or file setting column B (transfer destination). The display of the valid area (A or B) is highlighted.	C	6.2.1 Changing the Valid Area
Device select	This displays the submenu of the machining program storage area. When the submenu is selected, the device is confirmed, and if a directory exists it is set in the root. The memory is selected as the default.	A	6.2.2 Selecting a Device, Directory, and File
Dir	This menu sets the directory that carries out input/ output operations, and is on standby for input. Note that when memory is selected for the device, the directory can be selected from the submenu.	A	
File name	This menu sets the file name that carries out input/ output operations, and is on standby for input. When memory is selected for the device, setting is not necessary if the directory is not the program.	A	
List update	This updates the list. The list of the directly selected in the currently valid file setting column (A/B) is updated.	C	-
Transfr A→B	This copies the file in file setting column A (transfer origin) to the file setting column B (transfer destination). (The transfer origin file is not changed.) A message appears during transfer and when the transfer is completed.	B	6.2.3 Transferring a File
Compare A:B	This compares the files in file setting column A (transfer origin) and file setting column B (transfer destination).	C	6.2.4 Comparing Files (Compare)
Erase A	This erases the file in file setting column A. (Note) The NC memory (excluding programs), serial and Ethernet (host file) cannot be erased.	B	6.2.5 Erasing a File
Erase B	This erases the file in file setting column B. (Note) The NC memory (excluding programs), serial and Ethernet (host file) cannot be erased.	B	
Rename A→B	This changes the name of the file in file setting column A (transfer origin) to the name of the file in file setting column B (transfer destination). (Note) The same device must be selected for A and B. The NC memory (excluding programs) and serial cannot be renamed.	B	6.2.6 Changing a File Name
Comment nondisp	This changes whether to show or hide the comment field.	B	
Dir create	This creates a new directory in the directory of the currently valid file setting column (A/B). The directory can be created when HD, FD, memory card or DS is selected for the device.	A	6.2.7 Creating a Directory
Merge B→A	The file contents in the file setting column B are added to the file in the file setting column A. (The file in the file setting column B is not changed.) (Note) The NC memory (excluding programs), serial and Ethernet (host file) cannot be merged.	B	6.2.8 Merging a File
FD format	This formats the FD. This menu is only for 700 series.	A	6.2.9 Formatting an External Device
MemCrd format	The formats the front IC card.	A	
DS format	This formats the NC compact flash memory. This menu is only for 700 series.	A	
Warning cancel	This cancels a warning from network service.	C	6.2.13 Sharing Machining Data "Canceling a warning from network service"
Stop	This interrupts the process (transfer, compare, etc.) during execution.	C	-

6.2.1 Changing the Valid Area

When setting the file setting field A or B device, directory and file name on this screen, the area containing these must be valid.

The display area can be changed by pressing the menu key (Area change) or the cursor key  and . After changing, the data setting operation is valid in that area.



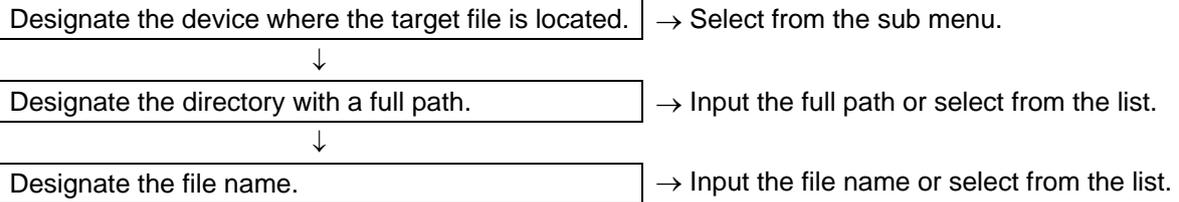
Changing the valid file setting field

When file setting field A (top) is valid

- (1) Press the menu .  The file setting field B (bottom) is validated.
 This can also be changed with the cursor key .

6.2.2 Selecting a Device, Directory and File

File selection sequence



Menu used

Device select menu's submenus

Menu	Details	Type	Reference
Memory	This selects the NC memory (program, parameter, PLC program, NC data).	C	-
HD	This selects the hard disk. This menu is only for 700 series.	C	-
Serial	This selects the RS-232C device (PC, tape, etc.).	C	-
Memory card	This selects the front IC card.	C	-
DS	This selects the NC compact flash memory. This menu is only for 700 series.	C	-
Ethernet	This selects the Ethernet-connected host computer.	C	-
FD	This selects the floppy disk. This menu is only for 700 series.	C	-
Anet server	This selects the Anshin-net server.	C	6.2.13 Sharing Machining Data

Dir (other than memory) and File name menu submenus

Menu	Details	Type	Reference
From list	The cursor appears in the list display. The list contents can be selected with the <input type="text" value="INPUT"/> key. When a directory is selected, the contents of the selected directory are displayed in the list. Continued selection is possible. When a file name is selected, the file name is temporarily displayed in the input area. When the <input type="text" value="INPUT"/> key is pressed again, it is fixed.	A	-

Selecting methods for device, directory, and file name

Device	Designation target file	Designation method		
		Device	Directory	File name
NC memory	<ul style="list-style-type: none"> Machining program User macro Fixed cycle 	Select from the submenu	- (Default)	Key input in the input area, and press <input type="text" value="INPUT"/> Select from the list
	Other than the machining program	Select from the submenu	Key input in the input area, and press <input type="text" value="INPUT"/> Select from the submenu	- (Fixed)
Other than the NC memory	All	Select from the submenu	Key input in the input area, and press <input type="text" value="INPUT"/>	Key input in the input area, and press <input type="text" value="INPUT"/>
			Select from the list	Select from the list

The device can be selected from the submenu. (The devices that can be used will differ depending on the specifications.)

One of the following methods can be used to designate the directory (for devices other than the NC memory) and file name.

- Set the directory path (full path) or file name in the input area, and press the key.
- Press submenu of the menu or . Move the cursor to the target directory or file name, and press the key.

A wild card (*) can be used when selecting a file name.

Notes when selecting a file

- During directory and file name setting, the designated directory, path or file name will be set, even if it does not actually exist. This will not cause an error. Note that the previously set directory is overwritten.
- When a file in the NC memory other than a machining program is designated, it is not necessary to set the file name. (The file name is fixed.)
- When a file name is selected from the menu, it first is displayed in the input area. However, at this time the file name has not yet been fixed. Press the key again to fix the file name.
- When the key is pressed when setting a file name, the file name in the input area is erased.
- When a fixed cycle program is designated, the basic common parameters "#1166 fixpro" must be set. Select "Memory" for the device, and "Program" for the directory.

Initial directory when the Ethernet is selected

An initial directory is specified at turning the power ON by the following parameter.

#9706 Host No.
 #9714 Host1 directory
 #9734 Host2 directory
 #9754 Host3 directory
 #9774 Host4 directory

Selecting an NC memory program

(1) Press the menu .

The following menu appears.

Memory	HD	Serial	Memory card	DS	Ethernet	FD	Anet server
--------	----	--------	-------------	----	----------	----	-------------

(When specifications of all devices is valid.)

(2) Press the menu .

"Memory" appears in the device name, and the default "Program" appears in the directory.

A:Dev	Memory
Dir:	Program
File	-

(3) Press the menu .

The following menu appears.

From list

<When inputting the file name from the input area>

(4) Input the file name
10013

A:Dev	Memory
Dir:	Program
File	10013
-	-

<When selecting the file name from the list>

(4)-1 Press the menu .

The cursor appears in the list.

<Program>	<Char>	<Comment>
101	73	MAIN
102	53	SUB1
103	54	SUB2
10011	519	
10012	139	
10013	100	

(4)-2 Move the cursor to directory and file name to be selected, and fix.

, ,

This operation is repeated until the cursor reaches the file.

When only the directory is selected by

key, select the file name separately.

(4)-3 Press the key.

The selected file name appears.

A:Dev	Memory
Dir:	Program
File	10013
-	-

Designating multiple files

(1) Designating multiple serial files

Multiple serial files can be transferred, compared and erased in the file setting column A. Set as follows in this case.

File : First file name
 - : Last file name



A:Dev	Memory
Dir:	Program
File	10013
-	10050

(2) Using a wild card

A wild card (*) can be used in the file name.

(Note) When serial or Anshin-net server is used, multiple files cannot be compared.

File : *
 -



All files will be selected.

A:Dev	Memory
Dir:	Program
File	*
-	

Selecting an NC memory file other than a program

(1) Press the menu .



The following menu appears.

Memory	HD	Serial	Memory card	DS	Ethnet	FD	Anet server
--------	----	--------	-------------	----	--------	----	-------------

(When specifications of all devices is valid.)

(2) Press the menu .



"Memory" appears in the device name, and "Program" appears as the default in the directory.

A:Dev	Memory
Dir:	Program
File	
-	

(3) Select the menu .



The following menu appears.

Program	Program all	Tool offset	Variabl	Coord offset	Tool life	Param	AUX param	System config
	Smp data						System param	Option

(4) Press the menu .



The directory and file name appear.

A:Dev	Memory
Dir:	Tool offset
File	TOOL.OFS
-	

(Note 1) The file name for each directory is fixed. Refer to "6.2.10 List of file names" for the file names.

(Note 2) When the menu or is selected, the setting of "File name" column is cleared.

Selecting a device file other than the NC memory (When separately selecting the directory and file name.)

(1) Press the menu .

The following menu appears.

Memory	HD	Serial	Memory card	DS	Ethnet	FD		Anet server
--------	----	--------	-------------	----	--------	----	--	-------------

(When specifications of all devices is valid.)

(2) Select a device.

The device name appears. The root directory is selected as the default.

A:Dev	HD
Dir:	/
File	
-	

(3) Select the menu .

The mode changes to the mode for inputting the directory name. The following menu appears.

From list

<When inputting the directory from the input area>

(4) Input the directory path as a full path.
/PRG/PRECU

<When selecting the directory from the list >

(4)-1 Press the menu .

The cursor appears in the list.

<Program>	<Char>	<Comment>
.	DIR	
..	DIR	
NCDATA	DIR	
PRG	DIR	

(4)-2 Move the cursor to directory to be selected, and fix.
, ,

The selected directory appears in the data setting column. The contents of the selected directory appear in the list.

A:Dev	HD
Dir:	/PRG/PRECU
File	
-	

Repeat this operation until the target directory is reached. When the target directory is reached, press the key and quit the mode for inputting the directory.

(Note) Refer to next page "Selecting a device file other than the NC memory (When simultaneously selecting the directory and file name.)" procedure (3) or later for method of file designation.

Selecting a device file other than the NC memory (When simultaneously selecting the directory and file name.)

(1) Press the menu .

The following menu appears.

Memory	HD	Serial	Memory card	DS	Ethnet	FD		Anet server
--------	----	--------	-------------	----	--------	----	--	-------------

(When specifications of all devices is valid.)

(2) Select a device.

The device name appears. The root directory is selected as the default.

A:Dev	HD
Dir:	/
File	
-	

(3) Press the menu .

The mode changes to the mode for inputting the file name.
The following menu appears.

From list

<When inputting the file name from the input area>

(4) Input the file name
10013.PRG

<When selecting the file name from the list>

(4)-1 Press the menu .

The cursor appears in the list.

<Program>	<Char>	<Comment>
.	DIR	
..	DIR	
10011.PRG	521 Oct 02 13:18 2002	
10012.PRG	141 Oct 02 13:19 2002	
10013.PRG	102 Oct 02 13:19 2002	
10014.PRG	163 Oct 02 13:20 2002	
10015.PRG	90 Oct 02 13:21 2002	

(4)-2 Move the cursor to directory and file name to be selected, and fix.
, ,

This operation is repeated until the cursor reaches the file.

When only the directory is selected and key is pressed, select the file name separately.

(4)-3 Press the key.

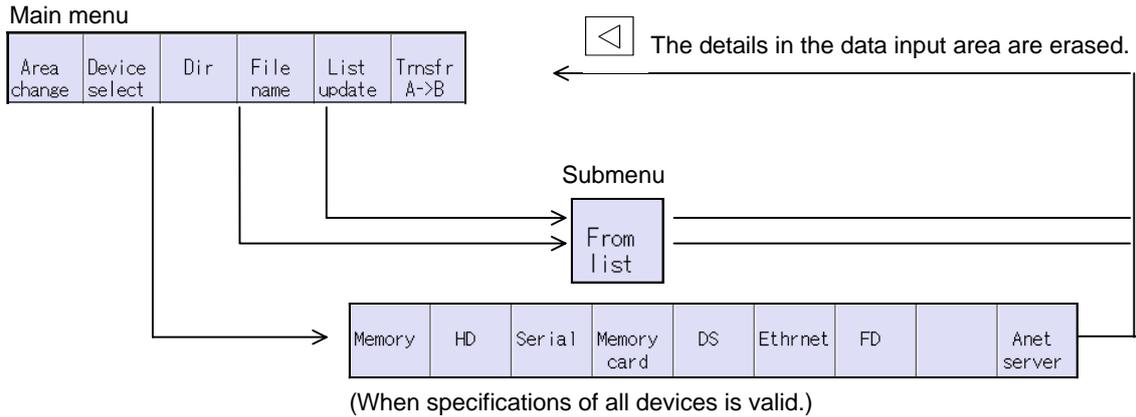
The selected file name appears.

A:Dev	HD
Dir:	/PRG/PRE CUT
File	10013.PRG
-	

Canceling the input mode

(1) Press the , or .
 To cancel the data input at this time, press the key.

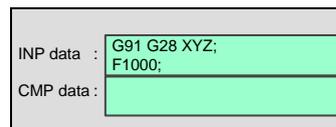
(2) Press the key.



6.2.3 Transferring a File

Operating method

- (1) Press the menu , and select file setting column A.
- (2) Designate the transfer origin device, directory and file name.  The designated file appears.
Multiple files can be designated in the file setting column A. Designate the first and last file name of the target range. A wildcard "*" can be designated for the file name.
- (3) Press the menu , and select file setting column B.
- (4) Designate the transfer destination device, directory and file name.  The designated file appears.
- (5) Press the menu .  A message appears to confirm the transfer.
- (6) Press or .  The file transfer starts. The data being transferred appears in the input data display column. A message appears when the transfer is completed.


 **Caution**

-  " ; ", "EOB", "%", and "EOR" are symbols used for explanation. The actual codes for ISO are "CR, LF" ("LF") and "%".
The programs created on the Edit screen are stored in the NC memory in a "CR, LF" format, however, the programs created with external devices such as the FD or RS-232C may be stored in an "LF" format.
The actual codes for EIA are "EOB (End of Block)" and "EOR (End of Record)".
-  To prevent the influence of data loss and data transformation over the line, always carry out data comparison after transferring a machining program.

Notes

(1) Notes related to transferring in general

- (a) Depending on the type of file, some data cannot be transferred during automatic operation. Do not transfer during automatic operation.
- (b) When the capacity of the transfer destination is reached during file transfer, only the data transferred up to that point is registered as a file, and an error will result.
- (c) During input to the NC memory or comparison, if the file format size on the NC memory side differs from the other side file format size (when the maximum number of registrations differs between the NC memory and the other side), processing is carried out matched to the smaller size.
(Ex. 1) If a format size of 200 files is input for a format size of 1000 NC files, 200 files are registered.
(Ex. 2) If a format size of 1000 files is input for a format size of 200 NC files, the files up to the 200th file are registered and an error message appears. (The remaining files are not registered.)
- (d) Up to 223 files, including the directory, can be registered in the FD's root directory.

(2) Notes when transferring machining program files

- (a) For the serial, always set feed (Null) at both ends of the "EOR" code at the head and end. If "EOB" etc., is directly after "EOR", the operation may not execute normally due to the input buffer influence during the next input operation.
- (b) The transfer speed is slower if there are many registrations.
- (c) The size of one block of the machining program should be 250 characters or less.
- (d) When using tape, carry out parity V adjustment to improve the reliability of the tape format. Then use with the input/output parameter "Parity V" validated.
- (e) When the machine tool builder macro and fixed cycle program are input to NC memory, change the program type with the parameter "#1166 fixpro". Also, set in the Input/Output screen as follows.
Device: Memory, Directory: Program
- (f) Transferring or verifying the multiple files between the external device connected serially and that other than the serial connection.
- (g) With machining program created before the MELDAS500 Series, "EOB" is registered as "LF". However, when these programs are stored in the 700/70 Series NC memory, "EOB" will be converted to "CR LF", and the number of characters will increase. Thus, when all of the machining programs output from an MELDAS500 Series or earlier NC, having the same specifications as the maximum memory capacity, are stored in the 700/70 Series NC memory, the memory capacity may be exceeded.
- (h) When the file to be transferred (input) is running or in "program restarting" mode, the operation message "Executing automatic operation" or "Program restarting" is displayed and file will not be transferred (input).

(3) Notes when transferring tool offset data files

- (a) If an error occurs during offset data transfer, an error message appears on the screen, and the transfer operation is interrupted.
- (b) Tool offset data file cannot be transferred from the serial to the memory.

(4) Notes when transferring parameter files

- (a) In the same manner as when setting in the Parameter screen, there are parameters validated immediately after input, and parameters validated after a restart. Restart when a parameter file has been transferred to the NC memory.
- (b) When a parameter file is transferred to the NC memory, the setting value of the input/output parameters is also changed. Before transferring next time, set the input/output parameters again.
- (c) System parameters can be transferred from the NC memory to an external device, but cannot be transferred from an external device to the NC memory.

(5) Notes when transferring common variable data files

- (a) If the variable value is 100000 or more or less than 0.0001 when transferring common variable data, it is expressed with an exponential expression.

(6) Notes when transferring tool life data files

- (a) When tool life data is output from the NC memory, the file information is inserted at the first and last of the file.
 First file information: Number of registered tools (P No.) and the maximum number of possible registrations (T No.)
 Last file information: Finish code

(7) Notes when transferring auxiliary axis parameter files (700 series only)

- (a) When the auxiliary axis parameters are input to the NC memory, the same parameter data is simultaneously transferred to the drive unit. If transferred errors by some causes occur, the parameter data may be not matched between NC memory data and drive unit.

(8) Notes when transferring sampling data file

- (a) When the output form is set as 8-digit hexadecimal number and the parameter "#1004 ctrlunit" is set to "E (1nm)", accurate data can be output just within 1m.
 When the output data length exceeds 1m, lower 32 bits of the sampling data will be output.

(9) Notes when transferring workpiece offset data files

- (a) Tool offset data file cannot be transferred from the serial to the memory.

6.2.4 Comparing Files (Compare)**Operation method**

- (1) Press the menu Area change, and select file setting column A.
- (2) Designate the device, directory and file name to be compared. ➔ The designated file name appears.
- (3) Press the menu Area change, and select file setting column B.
- (4) Designate the other side device, directory and file name to be compared. ➔ The designated file name appears.
- (5) Press the menu Compare A:B. ➔ The file comparison starts. The data being compared appears in the comparison data display column. A message appears when the comparison is completed.
 If a comparison error occurs, the block with the error is displayed in the comparison data display column on the screen.

```

INP data: (MAIN);G28XYZ;G00X10.;N1X20.
          ;X30.;X40.;N2X50.;M98P102H1;
CMP data: (MAIN);G28XYZ;G00X10.;N1X20.
          ;X30.;X40.;N2X50.;M98P103H1;
```

- (Note)** Files that can be compared are text files only.
 Correct outcome will not be obtained through binary file comparison.

6.2.5 Erasing a File

Erasing a file **Erase A**

- (1) Press the menu **Area change**, and select file setting column A.
- (2) Designate the device, directory and file name to be erased. → The designated file appears.
- (3) Press the menu **Erase A**. → A message to confirm the erasing appears.
- (4) Press **Y** or **INPUT**. → The file is erased.
A message appears when the erasing is completed. The number of programs registered, number of memory characters and remainder reappear. Press the **List update** menu to update the list.

(Note) If the file to be erased is running or in "program restarting" mode, the operation message "Executing automatic operation" or "Program restarting" is displayed, and the file will not be erased.

Erasing a file **Erase B**

The operations are the same as method "Erasing a file **Erase A**" above. Designate the target device, directory and file name in the file setting column B, and press the menu **Erase B**.

6.2.6 Changing a File Name (Rename)

Operation method

- (1) Press the menu **Area change**, and select file setting column A.
- (2) Designate the original device, directory and file name. → The designated file name appears.
- (3) Press the menu **Area change**, and select file setting column B.
- (4) Designate the new device, directory and file name. → The designated new file name appears.
- (5) Press the menu **Rename A -> B**.
- (6) Press **Y** or **INPUT**. → Renaming of the file starts.
A message appears when the renaming is completed.

(Note 1) Select the same device for original and new devices.

(Note 2) If the file to be renamed is running or in "program restarting" mode, the operation message "Executing automatic operation" or "Program restarting" is displayed, and the name will not be changed.

(Note 3) If a file that does not exist is designated for the original file, or if an existing file name is designated for the new file selection, an error occurs, and the name is not changed.

6.2.7 Creating a Directory

Operation method

- (1) Press the menu , and select file setting column A.
- (2) Designate the device.  The designated details appear.
The devices that can create a directory are HD, FD, Memory card and DS.
- (3) Designate the directory where the new directory is to be created.  The designated details appear.
- (4) Set the new directory in file setting column A.  The designated details appear.
- (5) Press the menu .  The designated directory is created.

The directory can also be created in file setting area B.

(Note 1) Up to 223 files, including the directory, can be registered in the FD's root directory.

6.2.8 Merging a File

Operation method

(1) Press the menu , and select the file setting column A.

(2) Designate the merge destination device, directory and file name.

➔ The designated file name appears.

(3) Press the menu , and select the file setting column B.

(4) Designate the merge origin device, directory and file name.

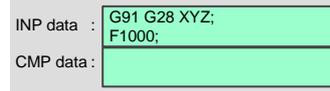
➔ The designated file name appears.

(5) Press the menu .

➔ A message confirming the merge appears.

(6) Press or .

➔ The file merge starts. The data being merged will appear in the Input Data display column. A message appears when the merge is completed.



The details of the file in file setting column A will be as follows when the merge is completed.

<pre>(FILE A) G28 XYZ; G90 F800; G00 X100. Y100.; (FILE B) G91 G28 XYZ; F1000; G01 X200. Y200.; M02; %</pre>	<p>} Details of file in file setting column A before merge.</p> <p>} Details of file in file setting column B before merge.</p>
--	---

The details of the file in file setting column B do not change.

(Note 1) If the merge destination file (file in file setting column A) is running or in "program restarting" mode, the operation message "Executing automatic operation" or "Program restarting" is displayed, and the files will not be merged.

(Note 2) The devices that can be merged are the memory, HD, FD, memory card and DS.

6.2.9 Formatting an External Device

Operation method (Formatting an FD) [700 series only]

- (1) Insert a floppy disk in the FD drive, and press **FD format**. → A message confirming the formatting appears.
- (2) Press **Y** or **INPUT**. → The FD is formatted.
A message appears when the formatting is completed.

(Note 1) The FD is formatted with FAT (1.44MB).

(Note 2) The volume label is set when formatting the FD.

Operation method (Formatting a memory card)

- (1) Press the menu **MemCrd format**. → A message confirming the formatting appears.
- (2) Press **Y** or **INPUT**. → The memory card is formatted.
A message appears when the formatting is completed.

(Note 1) The memory card is formatted with FAT16.

(Note 2) The volume label is set when the memory card is formatted.

Operation method (Formatting a DS) [700 series only]

First, press the menu **DS format**. Refer to "Formatting a memory card" for following operations.

(Note 1) The DS is formatted with FAT16.

(Note 2) Only the DS formatted with FAT16 can be used. The DS with NTFS cannot be used.

(Note 3) As for the DS formatted with NTFS, reformat it with FAT formatted by Windows to use.
(NC cannot convert NTFS partition to FAT formatted.)

(Note 4) The volume label is not set even when the DS is formatted.

6.2.10 List of File Names

There is a directory for each type of data in the NC memory.

Each directory and file name (fixed) in the NC memory is shown below.

Do not change the extensions (.XXX) when storing in a device other than the NC memory.

Data type	NC memory directory path	Fixed file name
Machining program	/PRG/USER	(Program No.)
Fixed cycle program	/PRG/FIX	(Program No.)
Machine tool builder macro	/PRG/MMACRO	(Program No.)
User macro	/PRG/UMACRO	(Program No.)
MDI	/PRG/MDI	(Program No.)
Parameters Parameters [User, machine] (Text format) Option file Auxiliary axis parameter (700 series only)	/PRM	ALL.PRM SYSTEM.PRM AUXAXIS.PRM
PLC program	/LAD	USERPLC.LAD
NC data Tool compensation amount data Tool life management data Tool management data Common variable data SRAM data Workpiece offset data	/DAT	TOOL.OFS TLIFE.TLF TOOLMNG.OFS COMMON.VAR SRAM.BIN WORK.OFS
System configuration data	/DGN	ASSEMBLY.INF
Backup file	/APLC	APLC.BIN
	/CUSTOM	CUSTOM.BIN (Only 70 Series)
Decryption code	/RLS	PASSCODE.DAT

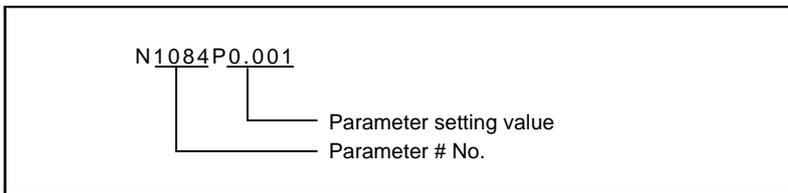
Data format (Parameter)

Parameter data format is as follows.

Address	Definition	Details
N	Parameter No.	The parameter No. is shown with the value following N.
A	Axis No.	For axis data, the axis No. is shown with the value following A. The first axis will be A1.
T	Part system No.	For data per part system, the part system No. is shown with the value following T. (\$1: T1, \$2: T2, PLC axis: T3)
C	Spindle No.	For spindle data, spindle No. is displayed with a numerical value following C. Thus, 1st spindle is displayed as C1.
K	Split data No.	For the parameter with which data is split and set, each split data is displayed. (Ex. IP address)
P	Parameter data	The parameter data is shown with the value following P.

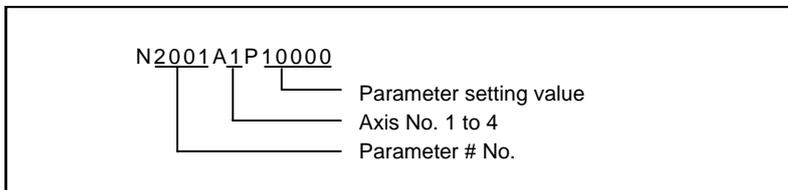
The following types of data format are used according to parameter type and display method.
(The address order in one block must use the following format.)

(1) Common parameter (One data item per one # No.)



(a) The output parameter setting value is the same format as the screen display.

(2) Axis parameter



(a) When multiple axes are displayed on one screen

The parameter data for the screen in which multiple axes are displayed on one screen is output per axis.

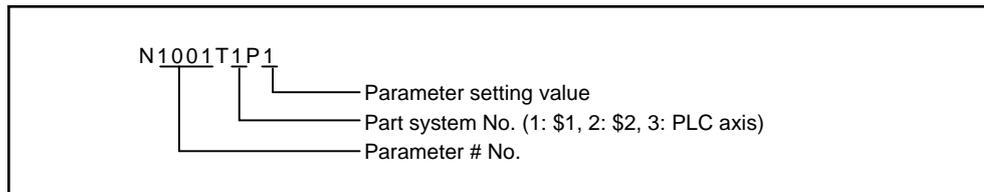
[Output example]

```

:
N2001A1P120000
N2002A1P4000
N2003A1P21
:
N2001A2P12000
N2002A2P4000
N2003A2P21
:
    
```

} 1st axis data

} 2nd axis data

(3) Part system parameter

- (a) The parameter data for the screen in which parameters per part system are displayed on one screen is out put as follows.

[Output example]

```

:
N1001T1P1
N1001T2P1
N1001T3P0
N1002T1P2
N1002T2P1
N1002T3P0
:

```

- (b) If parameters are displayed per part system upon switch of the part system (when key is pressed), the parameter data is output per part system.

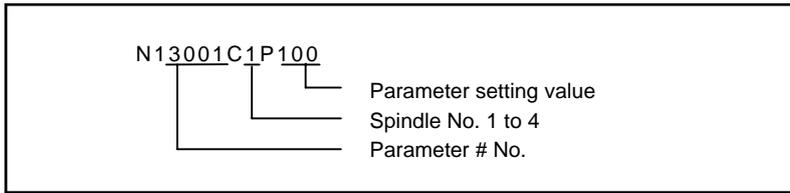
[Output example]

```

:
N8001T1P99
N8002T1P0
N8003T1P10000
:
N8001T2P30
N8002T2P1
N8003T2P20000
:

```

(4) Spindle parameter



(a) When multiple spindles are displayed in one screen

The parameter setting value for the screen in which multiple spindles parameters are displayed in one screen is output per spindle.

[Output example]

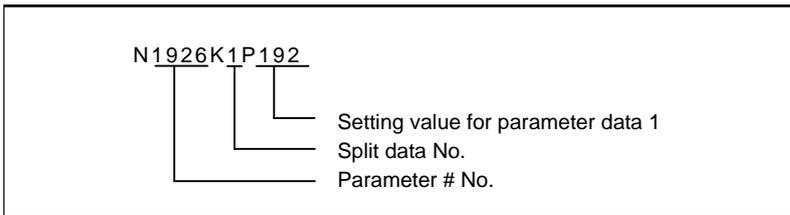
```

:
N13001C1P100
N13002C1P100
N13003C1P15
:
N13001C2P100
N13002C2P100
N13003C2P15
:
    
```

} 1st spindle data

} 2nd spindle data

(5) IP address



[Output example]

(Example) When "#1926 Global IP address" setting value is "192.168.200.1"

```

:
N1926K1P192
N1926K2P168
N1926K3P200
N1926K4P1
:
    
```

Data format (Tool compensation amount data)

(1) M type

Tool compensation amount data format is as follows.

Address	Definition	Details
G	-	Fixed to G10.
L	Type	L10 for tool compensation memory I. See below for tool compensation memory II. L10 : Tool length shape compensation L11 : Toll length wear compensation L12 : Tool radius shape compensation L13 : Tool radius wear compensation
P	Compensation No.	The compensation No. is shown with the value following P.
R	Compensation amount	The compensation No. is shown with the value following R.

[Output example]

```
%
$1
G10L10P1R0.000
G10L10P2R0.000
:
$2
G10L10P1R0.000
G10L10P2R0.000
:
%
```

(2) L type

Tool compensation amount data format is as follows.

Address	Definition	Details
G	-	Fixed to G10.
L	Type	L10 for tool length compensation input. L11 for tool nose wear compensation input.
P	Compensation No.	The compensation No. is shown with the value following P.
X	X axis compensation amount (Absolute)	The X axis compensation amount data is shown with the value following X. (Absolute)
U	X axis compensation amount (Incremental)	The X axis compensation amount data is shown with the value following U. (Incremental)
Z	Y axis compensation amount (Absolute)	The Y axis compensation amount data is shown with the value following Z. (Absolute)
W	Y axis compensation amount (Incremental)	The Y axis compensation amount data is shown with the value following W. (Incremental)
R	Nose R compensation amount (Absolute)	The nose R compensation amount data is shown with the value following R. (Absolute)
C	Nose R compensation amount (Incremental)	The nose R compensation amount data is shown with the value following C. (Incremental)
Q	Hypothetical tool nose point	The hypothetical tool nose point is shown with the value following Q.

[Output example]

```
%
G10L10P1X0.000Z0.000R1.000Q3
G10L11P1X0.000Z0.000R0.000Q3
G10L10P2X0.000Z0.000R1.000Q2
:
G10L11P80X0.000Z0.000R0.000Q0
%
```

Data format (Tool life management data)

(1) M type

Tool life management data format is as follows.

Address	Definition	Details
G	Group No.	The group No. for tool life management is shown with the value following G.
D	Tool no.	The tool No. is shown with the value following D.
S	Status	The tool status is shown with the value following S.
M	Method	The tool life management method is shown with the value following M.
L	Length compensation	The length compensation data is shown with the value following L.
R	Radius compensation	The radius compensation data is shown with the value following R.
B	AUX.	The auxiliary data is shown with the value following B.
E	Service lifetime	The service lifetime is shown with the value following E.
P	Used time	The used time is shown with the value following P.

[Output example]

```
%
LIFEDM()
$1
G100 D1000 S1 M101010 L100 R100 B11 E0 P0
G200 D2000 S2 M000000 L200 R200 B22 E0 P0
G300 D3000 S3 M201020 L300 R300 B33 E0 P0
:
G999999 D9999 S1 M202020 L999 R999 B99 E0 P0
%
```

(2) L type

(a) Tool life management I

Tool life management I format is as follows.

Address	Definition	Details
N	Tool No.	The tool No. is shown with the value following N.
P	Time-used	The time-used is shown with the value following P.
E	Time-max	The time-max is shown with the value following E.
Q	Count-used	The count-used is shown with the value following Q.
F	Count-max	The count-max is shown with the value following F.
S	Status	The status of tool life management is shown with the value following S.
B	AUX.	The auxiliary data is shown with the value following B.

[Output example]

```
%
LIFEL1( TOOL LIFE DATA)
$1
N1 P12.34.56 E45.56 Q6 F7 S1 B0
N2 P2.3.4 E5.6 Q7 F8 S1 B1
N3 P99.59.59 E99.59 Q9999 F9999 S2 B99
N4 P0.0.0 E0.0 Q0 F0 S0 B0
:
N80 P0.0.0 E0.0 Q0 F0 S0 B0
%
```

(b) Tool life management II

Tool life management II format is as follows.

Address	Definition	Details
G	Group No.	The group No. for tool life management is shown with the value following G.
M	Method	The tool life management method is shown with the value following M. 0: Life management is carried out using the time of cutting feed execution. 1: Life management is carried out using the number of usages. Counting the number of usages is executed when changing from rapid traverse command (G00, etc.) to cutting feed command (G01,G02,G03, etc.). Note that if rapid traverse command or cutting feed command that does not involve any movement is executed, usage will not be counted.
E/F	Service life	The life data (service lifetime) is shown with the value following E. The life data (service life count) is shown with the value following F. (Note 1) When the combination of the setting of tool life management method and address is illegal, error occurs at data input. (Note 2) The USED data will be incremented when the LIFE data is "0", but no judgment if the service life is reached will be made.
A/B	Remaining life	[Tool life prediction function valid (#1277 ext13 bit1 is "1")] The tool life prediction data (remaining lifetime) is shown with the value following A. The tool life prediction data (remaining life count) is shown with the value following B. (Note 1) When the value greater than the tool life data is set, tool life prediction signal is not output. (Note 2) When the combination of the setting of tool life management method and address is illegal, error occurs at data input. [Tool life prediction function invalid (#1277 ext13 bit1 is "0")] A and B are not used.
D	Tool No.	The tool No. is shown with the value following D.
H	Compensation No.	The compensation No. is shown with the value following H.
P/Q	Used	The used data (used time) is shown with the value following P or Q. The used data (used count) is shown with the value following P or Q. (Note 1) When the combination of the setting of tool life management method and address is illegal, error occurs at data input.
S	Status	The tool status is shown with the value following S. 0: Not used tool 1: Current tool (tool being used) 2: Normal life-reached tool 3: Tool skip tool

```
[Output example]
%
LIFEL2( TOOL LIFE DATA)
$1
G100 M0 E1000 A50 D123456 H1 P321 S1
G100 M0 E1000 A50 D888888 H2 P0 S0
G111 M1 F2000 B80 D777777 H20 Q2000 S2
G1234 M0 E1000 A50 D123 H30 P100 S1
G9999 M1 F999999 B9 D999999 H80 Q999999 S2
%
```

Data format (Common variable data)

Common variable data format is as follows.

Address	Definition	Details
N	Variable No.	The common variable # No. is shown with the value following N.
T	Part system No.	For data per part system, the part system No. is shown with the value following T. (\$1: T1, \$2: T2, PLC axis: T3)
P	Common variable data	The common variable data is shown with the value following P. An exponential is displayed when 7 digits are set in the integer section and 5 digits or more are set in the decimal section. (Example) 1234567 → 1.2346E+006, 0.00001 → 1.0000E-005

```
[Output example]
%
COMN()
N100T1P123.0000
N101T1P1.0000E7
:
N100T2P1.0000E7
:
%
```

Data format (System configuration data)

System configuration data format is as follows.

Contents of the data are same as that of displayed in the system configuration screen and drive monitor screen of the diagnosis screen.

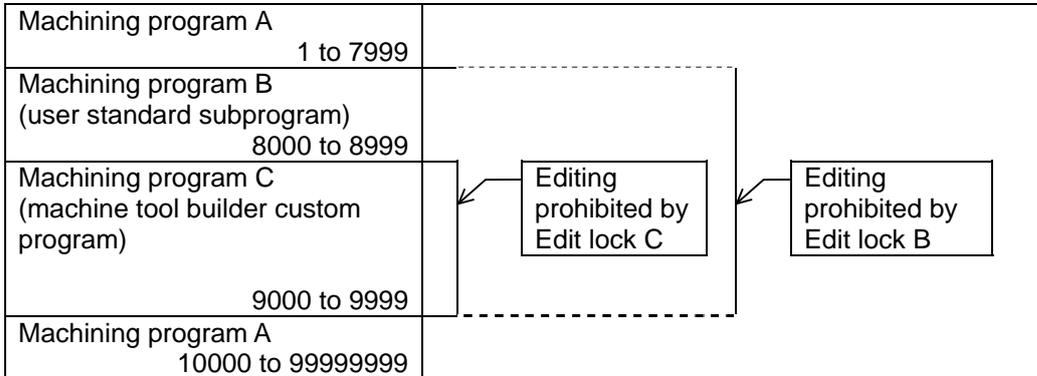
[COMMON] FILE=ASSEMBLY DATE=04.11.01 TIME=15:30:00 NC_TYPE=MITSUBISHI CNC 750M	
(1) NC unit configuration	Data & time of system configuration data output
(2) H/W configuration	NC type
(3) Servo drive unit information	Same as "H/W config" screen
(4) Spindle drive unit information	Same as "H/W config" screen
(5) Power supply unit information	Same as "Servo unit" screen
(6) Auxiliary axis information (700 series only)	Same as "Spindle unit" screen
(7) S/W configuration	Same as "Power unit" screen
(8) Ladder configuration	Same as "AUX unit" screen
[F_END]	Same as "S/W config" screen
	Same as "Ladder config" screen

[Output example]

[COMMON] FILE=ASSEMBLY DATE=04.11.01 TIME=15:30:00 NC_TYPE=MITSUBISHI CNC 750M	Tag described at the top of the file Header information of the file (file type) File output date File output time NC type and Lathe(L)/Machining center(M)
[FILE] TYPE=NC_UNIT NC_NAME=FCA750MN UNIT_NAME=FCA7-MA001 NC_SN=M1234567890 [END]	Tag described at the top of the data Data type (NC unit configuration) Describe items and data
[FILE] TYPE=HARDWARE [DATA1] HW_NAME=HN145 REVISION=A [DATA2] : [DATA n] HW_NAME=HN081 REVISION=A [END]	Tag described at the end of the data Data type (H/W configuration) Repetitive data No. tag (1st) Describe items and data Repetitive data No. tag (2nd) Repetitive data No. tag (n-th) Describe items and data
: [FILE] : [END] [F_END]	Tag indicating file end

6.2.11 Edit Lock B and C

This function prohibits editing, erasing, etc., of the machining programs B and C, and protects the machining programs in NC memory.



The operations below in the Edit MDI and the Input/Output screens are influenced by the edit lock setting. An error will result if operations that are not possible are attempted. When the edit lock is valid, processing is executed (except the edit lock target program) by the input/output function.

○ : Operation possible × : Operation not possible

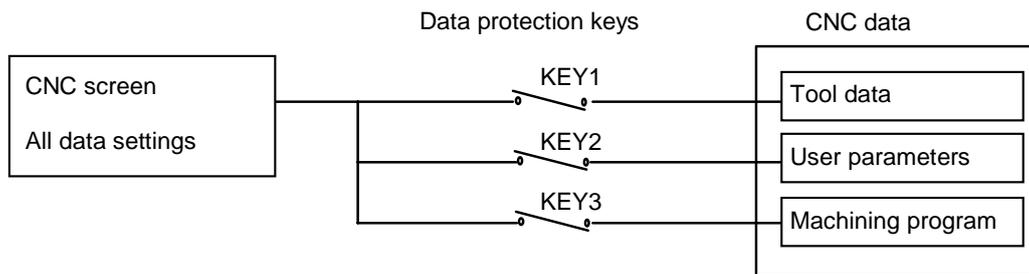
Screen	Operation	Edit lock B			Edit lock C		
		Machining program			Machining program		
		A	B	C	A	B	C
Edit	Search	○	○	×	○	○	×
	Edit	○	×	×	○	○	×
	MDI registration	○	×	×	○	○	×
Input/Output	Transfer	○	×	×	○	○	×
	Compare	○	×	×	○	○	×
	Copy	○	×	×	○	○	×
	Merge	○	×	×	○	○	×
	Rename	○	×	×	○	○	×
	Erase	○	×	×	○	○	×
Monitor	Buffer correction	○	×	×	○	○	×

6.2.12 Data Protect Keys

The data protect keys can be used to prohibit data setting and erasure. The following three keys are available. (Their names differ according to the machine tool builder. For further details, refer to the instruction manual issued by the machine tool builder.)

- (1) KEY1: Provides general protection for tool data and protection of the coordinate system preset using the origin set.
- (2) KEY2: Protects user parameters and common variables.
- (3) KEY3: Protects the machining program.

Data settings and deletion is stopped if any of the data protection keys is OFF.



(Note 1) When using the multi-part system, the settings for various types of data in all part systems are prohibited with the data protection keys.

Tool data protection (KEY 1)

The operations listed in the table below are prohibited when KEY 1 is OFF.

<Data protected by KEY1 >

No.	Operation	Screen
1	Origin set	Monitor
2	Tool compensation amount setting/erasing	Tool compensation amnt, Tool measure
3	Tool registration data setting/erasing	Tool registration
4	Tool life data setting/erasing	Tool life
5	Workpiece coordinate offset amount setting/erasing	Coord offset, Workpiece measure, Rotate meas
6	Tool offset data input/output	Input/Output
7	Tool compensation amount input/output	Input/Output
8	Tool registration data input/output	Input/Output
9	Tool life data input/output	Input/Output
10	Workpiece coordinate offset amount input/output	Input/Output
11	Tool management data input/output	Input/Output

(Note) If setting of data in the screens shown in No. 2 to 5 is attempted when the KEY1 is OFF, the message "Data protect" appears.

If origin set is attempted on the Position display1 screen and the key is pressed, the origin set is not executed, and the message "Origin set not possible" appears.

If the operations shown in No. 6 to 10 are attempted and the key is pressed, the message "Data protect" appears, and the input and output are disabled.

User parameter, common variable protection (KEY 2)

The operations listed in the table below are prohibited when KEY 2 is OFF.

<Data protected by KEY 2>

No.	Operation	Screen
1	Control parameter ON/OFF	Setup/User/Control param
2	Axis parameter setting	Setup/User/Axis param
3	Operation parameter setting	Setup/User/Operation param
4	Machining parameter setting	Setup/User/Process param, Rotate meas
5	Common variable setting	Monitr/Position display/Common variable
6	I/O basic parameter setting	Mainte/Parameter
7	Parameter input/output	Mainte/I/O
8	Common variable input/output	Mainte/I/O

(Note) If setting of data in the screens shown above is attempted when the KEY2 is OFF, the message "Data protect" appears.

If the operations shown in No. 7 and 8 are attempted and the key is pressed, the message "Data protect" appears, and the input and output are disabled.

Machining program protection (KEY 3)

The operations listed in the table below are prohibited when KEY 3 is OFF.

<Data protected by KEY3>

No.	Operation	Screen
1	MDI data memory entry	MDI
2	Machining program editing	Edit
3	New program editing	Edit
4	Registered program comment setting	Edit
5	Machining program memory registration, compare, input/output	Input/Output
6	Machining program erasing (single, all)	Input/Output
7	Registered program comment setting	Input/Output
8	Machining program copy, marge, name change	Input/Output
9	Machining program buffer correction	Monitr

(Note) If editing or setting, etc., is attempted on the screens shown above when the KEY3 is OFF, the message "Data protect" appears.

6.2.13 Sharing Machining Data

Machining data can be send to/received from Anshin-net server in the call center.

Operation method

- | | |
|--|---|
| (1) Use the menu <input type="text" value="Area change"/> to select file setting column A. | |
| (2) Designate the transfer source device, directory and file name. | ➔ The designated file appears. |
| (3) Use the menu <input type="text" value="Area change"/> to select file setting column B. | |
| (4) Press the menu <input type="text" value="Anshin net"/> to designate the transfer destination device and file name. | ➔ "Anshin-net server" is displayed as device name. The file name, unless specified, is as same as the name of the originally transferred file. Only numbers within 10 characters can be used for a file name. |
| (5) Press the menu <input type="text" value="Transfr A→B"/> . | ➔ A message appears to confirm the transmission of machining data. |
| (6) Press the <input type="text" value="Y"/> or <input type="text" value="INPUT"/> key. | ➔ Transmission of machining data starts. A message appears upon the completion of transmission. |

(Note 1) Neither HD operation nor memory card operation can be performed during file transmission from HD.

(Note 2) Only one file can be saved on Anshin-net server. The file previously exists on the Anshin-net server is overwritten.

(Note 3) Anshin-net icon appears in the upper middle of the screen during Anshin-net communications.

(Note 4) When Anshin-net server is specified as device name, the directory button is displayed in gray and cannot be selected.

Receiving machining data from Anshin-net server

- (1) Use the menu to select file setting column A.
- (2) Press the menu to designate the transfer source device and file name. → "Anshin-net server" as device name and the designated file name are displayed.
- (4) Use the menu to select file setting column B.
- (5) Designate the transfer destination device, directory and file name. → The designated file appears.
- (6) Press the menu . → A message appears to confirm the reception of machining data.
When the file name is not designated in the file setting column A, the message "File name not designated for dev A" appears to indicate the error.
- (7) Press the or key. → The reception of machining data starts.
A message appears upon the completion of reception.

(Note 1) When the same file name exists in the transfer destination device, an error occurs and the transfer cannot be executed.

(Note 2) Anshin-net icon appears in the upper middle of the screen during Anshin-net communications.

(Note 3) When Anshin-net server is specified as device name, the directory button is displayed in gray and cannot be selected.

Erasing machining data on Anshin-net server

- (1) Use the menu to select file setting column A.
- (2) Press the menu to designate the file name to erase. → "Anshin-net server", as device name, and the designated file name are displayed.
- (3) Press the menu → A message appears to confirm the erasing.
- (4) Press the or key. → The file will be erased.
A message appears upon the completion of erasing.

Canceling a warning from network service

- (1) Press the menu key → The warning message from network service is cleared.

(Note 1) A warning message from network service appears when any error occurs during communications.

(Note 2) NC reset can also clear the warning message from network service.

Notes

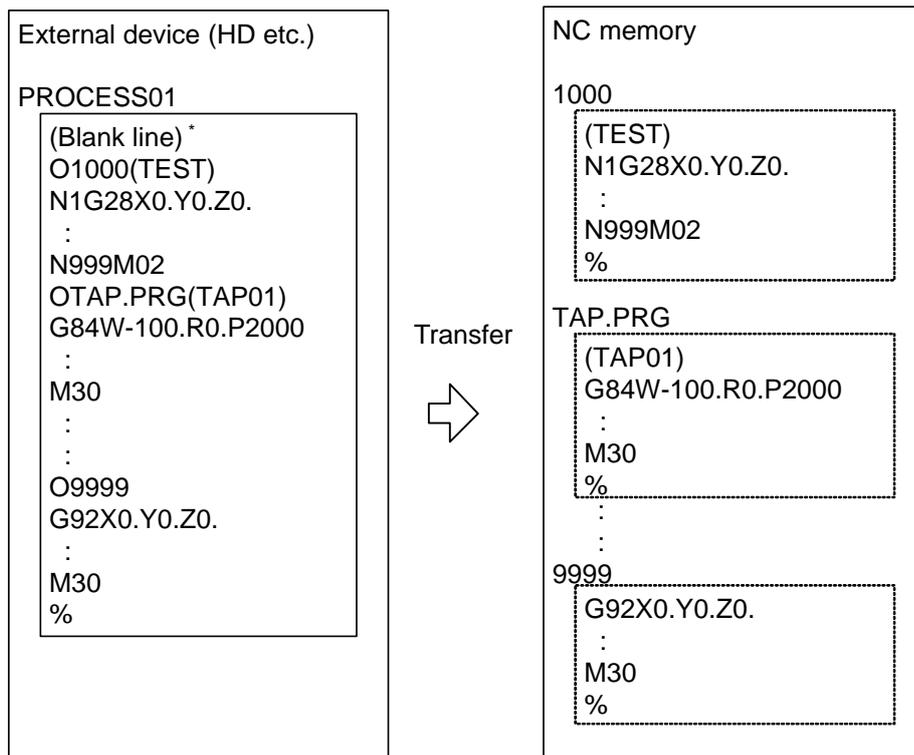
- (1) Transfer to/from Anshin-net server cannot be interrupted.
- (2) Only machining program, except fixed cycle and machine tool builder macro, can be transferred to Anshin-net server.
- (3) Data transfer between Anshin-net server and serial (RS232C) or Ethernet is not available.
- (4) The edit screen and the maintenance screen can share machining data. However, the machining data being transferred to/from either of the screens cannot be shared with the other screen.
- (5) Wildcard (*) cannot be used for the file name.
- (6) The capacity of machining program for transmission is restricted according to the settings of Anshin-net server.

6.2.14 The Batch Input/Output the Machining Program of NC Memory

One file which consists of two or more machining program can be transferred to NC memory by dividing the file. The machining programs united with one file can be transferred to the external device. The targets are the machining programs of the user.

Batch input and verify to NC memory

Example of "the external device -> NC memory" of machining program batch transfer



When the external device is serial, the top line is %.

When the transfer direction file name is designated, the top program name of transfer source can be omitted. ("O1000" can be omitted in above example.)

- (1) Press the menu to select file setting column A.
- (2) Designate the transfer source device and directory. ➡ The designated device and directory appear.
- (3) Designate the file which consists of two or more machining programs. ➡ The designated file name appears.
- (4) Press the menu to select file setting column B.
- (5) Designate the transfer direction device (memory). ➡ The designated device name (memory) appears.

- (6) Press the menu Dir - Program all.

➔

The "Program all" appears in the directory.

When the file name is designated, only the top program of transfer source is stored in the designated file.

For example of "Example of "the external device -> NC memory" of machining program batch transfer", when "TESTCUT" is designated as the transfer direction file name, the top program is transferred as not "1000" but "TESTCUT".
- (7) Press the menu Transfr A→B.

➔

The message "OK? (Y/N)" appears.
- (8) Press the Y or INPUT key.

➔

Transmission of machining data starts. Transmitting data is displayed in input data display column. The message "Transfer complete" appears upon the completion of transmission.

When the same machining program already exists in transfer direction device, the message "Overwrite this file?(Y/N)" appears. The transferring the following program is continued by selecting Y or N key.
- (9) Press the menu Compare A:B key.

➔

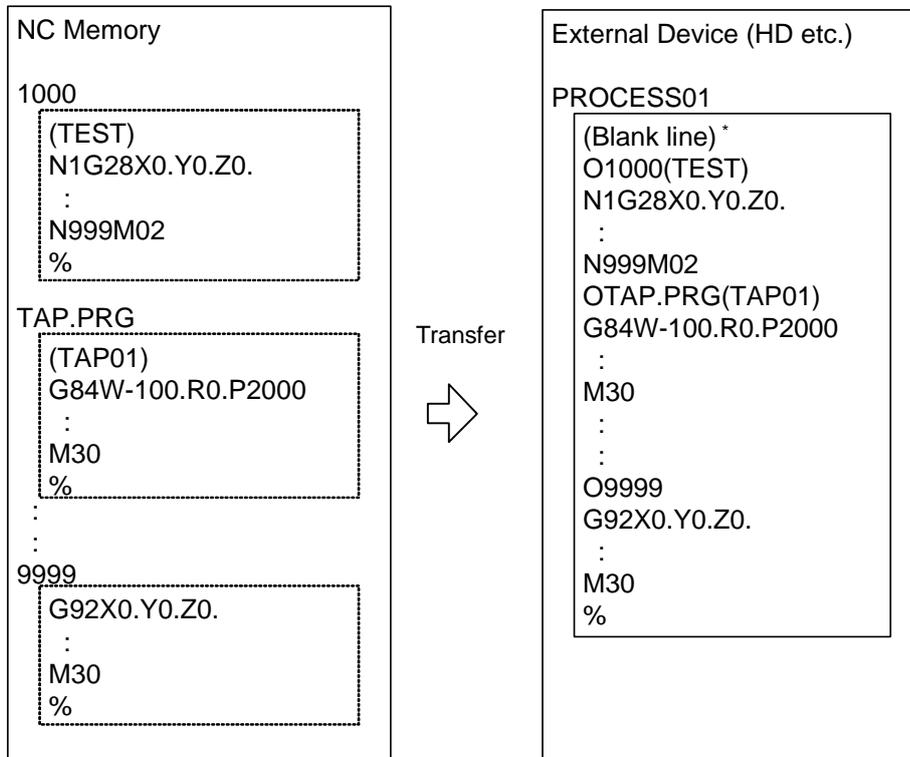
The verification of the file starts. The verifying data is displayed. When the verification is completed, the message appears. When the verification error occurs, the error block is displayed in verification data display column and the message "Compare error. Compare next file?(Y/N)" appears.

- (Note 1)** When programs are input in a batch to NC memory, always specify "Program batch" to NC memory directory as transfer direction. If "Program batch" is not specified, one file is transferred (No batch).
- (Note 2)** When the program which is subject to edit lock exists in transfer source, the message "Edit lock B" or "Edit lock C" appears, and transfer is interrupted. When the program which is not subject to edit lock is transferred, delete the program which is subject to edit lock in transfer source. (Refer to "6.2.11 Edit Lock B and C".)
- (Note 3)** When the machining program protection is valid (KEY3 is OFF), the program cannot be transferred/verified. (Refer to "6.2.13 Data Protect Keys".)
- (Note 4)** The program during automatic operation, program restart or program checking cannot be overwritten. the message "Executing automatic operation", "Program restarting" or "Program checking" appears, and transfer is interrupted.
- (Note 5)** The first line of the transfer source file is ignored.
- (Note 6)** When the transfer source is serial, the program is similarly transferred even if direction of following case 1 and case 2 is set.

	Case 1	Case 2
Device name	Memory	Memory
Directory	Program batch	Program
File name	(Null or file designation)	(Null or file designation)

Batch output and verify from NC memory

Example of "NC memory -> the external device" of machining program batch transfer



When the external device is serial, the top line is %.

- (1) Press the menu to select file setting column A.
- (2) Designate the transfer source device (memory). The designated device (memory) appears.
- (3) Press the menu - . The "Program all" appears in the directory.
Do not specify the file name. If the name is specified, an error will occur.
- (4) Press the menu key to select file setting column B.
- (5) Designate the transfer direction device and directory. The designated device name and directory appear.
- (6) Designate the transfer direction file to input multiple machining programs. The designated file name appears.
When the file name is not designated, output to the file "ALL.PRG" of the designated directory.

(7) Press the menu .



The message "OK? (Y/N)" appears.

(8) Press the or key.



Transmission of machining data starts. Transmitting data is displayed in input data display column. The message "Transfer complete" appears upon the completion of transmission.

When the same machining program already exists in transfer direction device, the message "Overwrite this file?(Y/N)" appears. The transferring the following program is continued by selecting or key.

(9) Press the menu key.



The verification of the file starts. The verifying data is displayed. When the verification is completed, the message appears. When the verification error occurs, the error block is displayed in verification data display column and the message "Compare error. Compare next file?(Y/N)" appears.

- (Note 1)** When programs are output in a batch from NC memory, always specify "Program batch" to NC memory directory as transfer source. If "Program batch" is not specified, the program is transferred by one program per one file.
- (Note 2)** When the programs which are subject to edit lock exists in NC memory, those file is not transferred. (The files are transferred excluding those files.) The verification is executed excluding the programs which are subject to edit lock. (Refer to "6.2.11 Edit Lock B and C".)
- (Note 3)** When the machining program protection is valid (KEY3 is OFF), the program cannot be transferred/verified. (Refer to "6.2.13 Data Protect Keys".)
- (Note 4)** When the transfer direction is serial, the program is similarly transferred even if source of following case 1 and case 2 is set.

	Case 1	Case 2
Device name	Memory	Memory
Directory	Program batch	Program
File name	(Null)	* (Wild card)

Notes

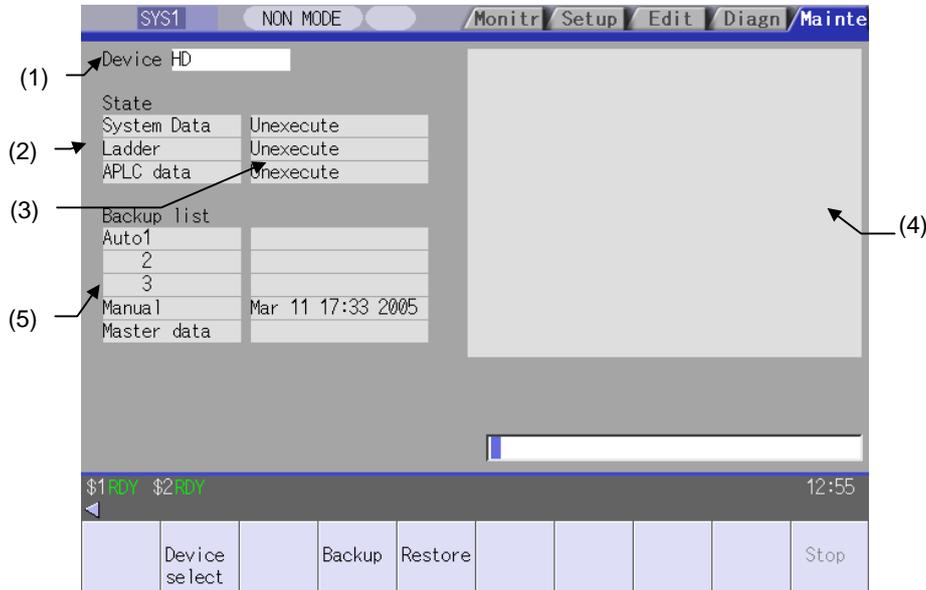
- (1) When the machining program batch input/output function of NC memory is used, "(" cannot be used for the machining program name. It is recognized that the inside of "(" is a comment.
- (2) The file name of file which consists of two or more machining program can have up to 32 characters, including the extension as well as the machining program.
- (3) When the "Program batch" is designated to NC memory directory, it is not possible to delete, rename and merge files. When the files are deleted in a batch, use "Program" for directory, and a wild card (*) for file name.
- (4) Refer to Note of "6.2.3 Transferring a File" for other notes.

6.3 All Backup Screen

This screen is used to perform batch backups of NC memory data to an external device, and batch restoration of that data to the NC memory.

Data backed up by the automatic backup function can also be restored.

Data which has been automatically backed up can be selected only when the device set by the "#8919 Auto backup device" parameter setting is selected.



Display items

Display item	Details												
(1) Device name	This displays the selected device name.												
(2) Data name	This displays the data name being backed up/restored. System data, ladder, APLC data and custom data (70 Series only) are displayed.												
(3) Execution status	This displays the processing execution status. Processing is executed in the system data, ladder, and APLC data order. (Note 1)												
(4) Warning message	This displays messages at the start and end of backup/restore processing.												
(5) Backup list	This displays the backup date list. This date is the system data time stamp. <table border="1" style="margin: 10px auto;"> <thead> <tr> <th>Backup area</th> <th>Explanation</th> <th>Backup format</th> </tr> </thead> <tbody> <tr> <td>Auto 1 2 3</td> <td>Data that was automatically backed up. Display in the backup date descending order. The latest three generations of data are displayed.</td> <td>Auto</td> </tr> <tr> <td>Manual</td> <td>Data that was backed up on this screen.</td> <td>Manual</td> </tr> <tr> <td>Master data</td> <td>Data that was backed up on this screen. Generally, this is the factory settings data.</td> <td>Manual</td> </tr> </tbody> </table> <p>The above data can be selected at restore processing. (Note 2)</p>	Backup area	Explanation	Backup format	Auto 1 2 3	Data that was automatically backed up. Display in the backup date descending order. The latest three generations of data are displayed.	Auto	Manual	Data that was backed up on this screen.	Manual	Master data	Data that was backed up on this screen. Generally, this is the factory settings data.	Manual
Backup area	Explanation	Backup format											
Auto 1 2 3	Data that was automatically backed up. Display in the backup date descending order. The latest three generations of data are displayed.	Auto											
Manual	Data that was backed up on this screen.	Manual											
Master data	Data that was backed up on this screen. Generally, this is the factory settings data.	Manual											

(Note 1) "APLC data" cannot be backed up/restored if the optional "APLC" is disabled.

(Note 2) The "Auto 1 to 3" data display when the device set by the "#8919 Auto backup device" parameter is selected.

Menus

Menu	Detail	Type	Reference
Device select	This displays the sub-menu for "Device" selection.	C	
Backup	This executes backup processing.	A	6.3.1 Performing a Backup Operation
Restore	This executes restore processing.	A	6.3.2 Performing a Restore Operation
Stop	This stops processing.	C	

6.3.1 Performing a Backup Operation

Operation methods

- (1) Press the menu . → The menu is highlighted.
An operation message "Select directory to backup" appears.
- (2) Move the cursor to select the area.
Press the key. → An operation message "OK? (Y/N)" appears.
(Note) Auto 1 to 3 cannot be selected.
- (3) Press or key. → Backup operation begins.
An operation message "Backopping" appears.
- When the backup is completed without error, an operation message "Backup complete" appears.

6.3.2 Performing a Restore Operation

Operation methods

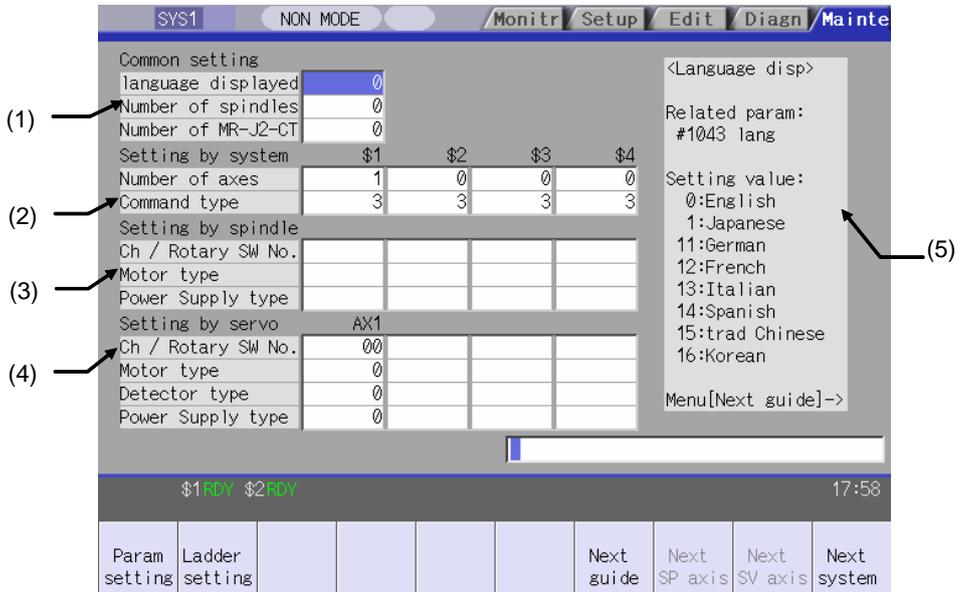
- (1) Press the menu . → The menu is highlighted.
An operation message "Select directory to restore" appears.
- (2) Move the cursor to select the file.
Press the key. → An operation message "OK? (Y/N)" appears.
- (3) Press or key. → Restore operation begins.
An operation message "Restoring" appears.
- When the restoration is completed without error, an operation message "Restore complete" appears.

6.4 System Setup Screen

The essential data required for the NC's initial startup is automatically set by inputting the minimum requirement item only on this screen.

This function allows setting data required to drive the motors (servomotors, spindle motor, auxiliary axis motor) to be specified quickly and easily, resulting in a faster initial startup of the tool machine.

(Note) The auxiliary axis motor is used for 700 series only.



Display items

Display item	Details
(1) Common setting area	
Language displayed	Set the display language for system setup and after setup (after the power is turned ON again). This setting is registered at the "#1043 lang".
Number of spindles	Set the number of spindles connected to the NC. This setting is registered at the "#1039 spinno".
Number of MR-J2-CT (700 series only)	Set the number of auxiliary axes (MR-J2-CT) connected to the NC. This setting is registered at the "#1044 auxno".
(2) Setting by system area	
Number of axes	Set the number of axes for each part system and PLC. This setting is registered at the "#1002 axisno". A setting error occurs if a setting of "1 or higher" is specified for the Nos.2 to 4 part systems when the setting for the previous part system is "0".
Command type	Set the command type for each part system. This setting is registered at the "#1037 cmdtyp". (Note) Although this setting can be set individually for each part system, it will be shared by the entire part system if specified for the machining center.

Display item	Details
(3) Setting by spindle area	
Ch / Rotary SW No.	Set the servo I/F connection channel and the rotary switch No. (2-digit value) for each spindle drive unit. This setting is registered at the "#3031 smcp_no". 1st digit : Servo I/F connection channel. 2nd digit : Rotary switch No.
Motor type	Set the motor types that are connected to each spindle. Input the values as indicated at the guidance display area. The input values are not converted to motor types.
Power Supply type	Set the power supply types that are connected to each spindle drive unit. Input the values as indicated at the guidance display area. The input values are then converted to, and display as, power supply types. If "0" is displayed, it means "No connection".
(4) Setting by servo area	
Ch / Rotary SW No.	Set the servo I/F connection channel and the rotary switch No. (2-digit value) for each servo drive unit. This setting is registered at the "#1021 mcp_no". 1st digit : Servo I/F connection channel. 2nd digit : Rotary switch No.
Motor type	Set the motor types that are connected to each servo axis. Input the values as indicated at the guidance display area. The input values are then converted to, and displayed as, motor types.
Detector type	Set the encoder types that are connected to each servomotor. Input the values as indicated at the guidance display area. The input values are then converted to, and displayed as, encoder types.
Power Supply type	Set the power supply types that are connected to each servo drive unit. Input the values as indicated at the guidance display area. The input values are then converted to, and displayed as, power supply types. If "0" is displayed, it means "No connection".
(5) Guidance display area	Indicate the setting content and setting range for the setting item where the cursor is positioned.

Menus

Menu	Details	Type	Reference
Param setting	This writes the parameters based on settings on system setup screen to NC system.	A	6.4.2 Writing Initial Parameters
Ladder setting	This writes the minimum required setup PLC ladder for manual operation (JOG mode) to NC system. This menu is only for 700 series.	A	6.4.3 Writing Sample PLC Ladders
Next guide	This displays the next page when guidance display content extends through multiple pages.	C	/
Next SP axis	This displays the 5th axis and following in setting by spindle area when the total number of axis in the part system exceeds 5.	C	
Next SV axis	This displays the 5th axis and following in setting by servo area when the total number of axis in the part system exceeds 5.	C	
Next system	This displays the PLC axis setting area at the setting by system area.	C	

6.4.1 Preparation for Spindle Parameter Setting

There are following two methods of setting spindle parameters.

(1) A spindle parameter setting file (INITSP.PRM) is prepared in advance.

(2) A spindle type is set by using this screen.

The following explanation is "Method of preparing a spindle parameter setting file in advance."

Operation methods (When a spindle parameter setting file is prepared)

- (1) Save the spindle parameter setting file at the memory card's root directory.

➔

Assuming that personal computer recognizes the memory card as the E drive, the spindle parameter setting file is saved as "E: \INITSP.PRM".

- (2) Insert the memory card into the front IC card holder.

➔

At the input/output screen, verify that the "INITSP.PRM" file has been saved on the memory card.

6.4.2 Writing Initial Parameters

Operation methods

- (1) Enter all the required settings on the system setup screen.
- (2) Press the menu . → The menu is highlighted. An operation message "Set up parameter? (Y/N)" appears.
- (3) Press the or key. → Parameter setting begins, and an operation message appears as follows. "Initial parameter creating" → "Initial parameter transferring" → "Spindle initial parameter transferring" (This operation message is displayed only if a spindle parameter setting file exists.) When parameter settings are completed, an operation message "Param set ended. Format NC memory? (Y/N)" appears.
- (4) Press the or key. → File formatting begins, and an operation message "Executing format" appears. When file formatting is completed, an operation message "Format complete" appears.

(Note 1) When the "data protect key 2" is enabled, an operation message "Data protect" appears, and processing is interrupted.

(Note 2) In the automatic operation mode, an operation message "Executing automatic operation" appears, and processing is interrupted.

(Note 3) If all the required settings have not been input on the system setup screen, an operation message "Setting error" appears, and processing is interrupted.

(Note 4) Processing is interrupted if a key other than the or key is pressed, or if switching to another screen occurs. (Processing is interrupted even if switching to another screen occurs during data transfers.)

(Note 5) The menu is displayed in gray during data transfers.

(Note 6) When the inserted memory card contains the spindle parameter setting file (INITSP.PRM), the default parameters of "Setting by spindle" set on the screen are overwritten with those from the file.

6.4.3 Writing Sample PLC Ladders

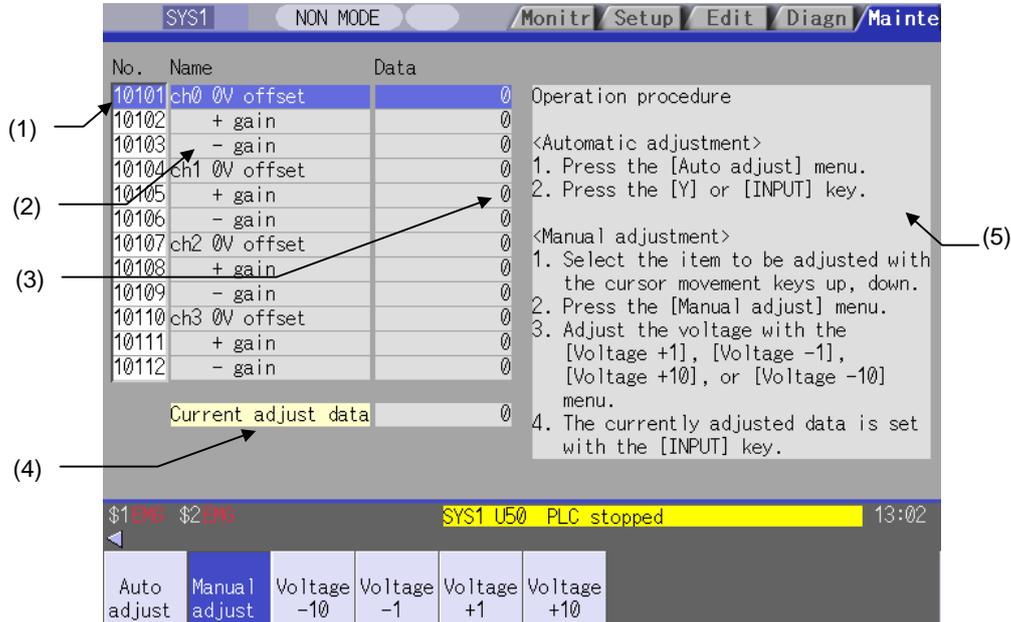
Operation methods

- (1) Press the menu .  An operation message "Write sample ladder? (Y/N)" appears.
- (2) Press the or key.  Sample PLC ladder writing begins, and an operation message "Transferring" appears. When sample PLC ladder writing is completed, an operation message "Transfer complete" appears.

- (Note 1)** In the automatic operation mode, an operation message "Executing automatic operation" appears, and processing is interrupted.
- (Note 2)** In the "PLC RUN" mode, an operation message "PLC running" appears, and processing is interrupted.
- (Note 3)** If there is no sample PLC ladder, an operation message "Sample ladder not found" appears, and processing is interrupted.
- (Note 4)** Processing is interrupted if a key other than the or key is pressed, or if switching to another screen occurs. (Processing is interrupted even if switching to another screen occurs during data transfers.)
- (Note 5)** The menu is displayed in gray during data transfers.

6.5 Adjust S-analog Screen

The spindle analog output adjustment is carried out on this screen.



Display items

Display item	Details
(1) Parameter No.	For manual adjustments, this uses the <input type="button" value="↑"/> <input type="button" value="↓"/> keys to move the cursor to the setting item to be adjusted. The cursor does not display during automatic adjustments.
(2) Adjustment item name	This displays the names of the spindle analog adjustment items.
(3) Adjustment data	This displays the setting data for each adjustment item.
(4) Current adjust data	This displays data which is being adjusted manually. (Display only during a manual adjustment.)
(5) Operation procedure	This displays the operation procedures for automatic and manual adjustments.

Menus

Menu	Details	Type	Reference
Auto adjust	This carries out automatic adjustment of the spindle analog output.	A	6.5.2 Performing Automatic Adjustments
Manual adjust	This carries out manual adjustment of the spindle analog output.	B	6.5.3 Performing Manual Adjustments
Voltage -10	This is used during manual adjustments to adjust downward in decrements of 10.	C	
Voltage -1	This is used during manual adjustments to adjust downward in decrements of 1.	C	
Voltage +1	This is used during manual adjustments to adjust upward in increments of 1.	C	
Voltage +10	This is used during manual adjustments to adjust upward in increments of 10.	C	

6.5.1 Adjustment Preparations

Operation methods

The PLC must be stopped when performing spindle analog output adjustments.

- | | | | |
|-----|---|---|-----------------------------------|
| (1) | Press the function key MAINTE . | ➔ | The maintenance screen appears. |
| (2) | Press the menu Psswd input . | ➔ | The menu is highlighted. |
| (3) | Enter the machine tool builder password. | ➔ | The menu highlighting is cleared. |
| (4) | Press the menu PLC STOP . | ➔ | The PLC is stopped. |

6.5.2 Performing Automatic Adjustments

Automatic adjustment will execute "0V offset adjustment" and "+ gain adjustment" for the remote I/O analog outputs (max. 4ch) connected to the main unit. The - gain will be handled as the + gain.

Operation methods

- | | | | |
|-----|---|---|--|
| (1) | Press the menu Auto adjust . | ➔ | The menu is highlighted.
An operation message "Execute? (Y/N)" appears.
The cursor display clears from the screen. |
| (2) | Press the Y or INPUT key. | ➔ | Automatic adjustment begins, and an operation message "Auto adjust execution" appears. The manual adjustment related menu is displayed in gray during the automatic adjustment, and manual adjustments are disabled. Switching to other screens is also disabled at this time.
When the automatic adjustment is completed, an operation message "Auto adjust complete" appears. |

6.5.3 Performing Manual Adjustments

Adjusting the ch0 0V offset

- (1) Use the keys to move the cursor to item #10101, then press the menu button. → The menu and the selected line are highlighted. The current setting value is displayed at "Current adjust data". A 0V analog voltage is output to ch0 at this time.
- (2) Monitor the analog voltage. → Adjust the 0V offset until the analog voltage is 0V ±3.5mV. The setting range -999 to 999.
- (3) <For plus-voltage>
Press the menu or . → "Current adjust data" is adjusted downward in decrements of 1 each time the menu is pressed.
"Current adjust data" is adjusted downward in decrements of 10 each time the menu is pressed.
- <For minus-voltage>
Press the menu or the . → "Current adjust data" is adjusted upward in increments of 1 each time the menu is pressed.
"Current adjust data" is adjusted upward in increments of 10 each time the menu is pressed.
"Current adjust data" range is -999 to 999.
- (4) Press the key. → "Current adjust data" is set as "ch0 0V offset", and the 0V offset result is enabled. The menu highlighting is cleared.

Adjusting the ch0 + gain

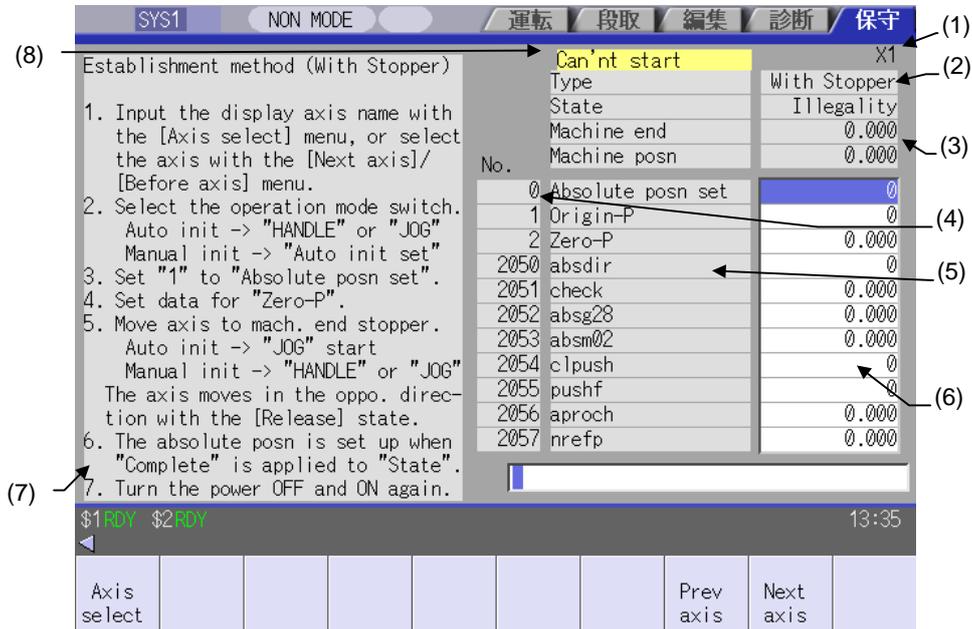
- (1) Use the keys to move the cursor to item #10102, then press the menu . → The menu and the selected line are highlighted. The current setting value is displayed at "Current adjust data". A ch0 +10V analog voltage is output at this time.
- (2) Monitor the analog voltage. → Adjust the + gain until the monitor voltage is 10V ±3.5mV. The setting range -999 to 999.
- (3) <For 10V or higher>
Press the menu or the .
- <For less than 10V>
Press the menu or the .
- (4) Press the key. → "Current adjust data" is set as "ch0 + gain", and the adjustment result is enabled. The menu highlighting is cleared.

Adjusting the ch0 - gain

- (1) Use the keys to move the cursor to item #10103, then press the menu . → The menu and the selected line are highlighted. The current setting value is displayed at "Current adjust data". A ch0 -10V analog voltage is output at this time.
- (2) Monitor the analog voltage. → Adjust the - gain until the monitor voltage is -10V ±3.5mV. The setting range -999 to 999.
- (3) <For -10V or higher>
Press the menu or .
- <For less than -10V>
Press the menu or .
- (4) Press the key. → "Current adjust data" is set as "ch0 - gain", and the adjustment result is enabled. The menu highlighting is cleared.

6.6 Absolute Position Setting Screen

The Absolute position setting screen is used to set the data for the absolute position of servo axes including arbitrary NC axes and PLC axes.



Display items

Display item	Details																						
(1) Axis name	This displays the axis name set with the parameter "#1022 axname2". The axis name can be switched by <u>Axis select</u> , <u>Next axis</u> , or <u>Prev axis</u> .																						
(2) Type of zero point initialization	<p>This displays the type of zero point initialization for the selected axis. The shortened expression of the zero point initialization method selected with the parameter "#2049 type (type of absolute position detection system)".</p> <table border="1"> <thead> <tr> <th></th> <th>Displayed expression</th> <th>Type of zero point initialization for absolute position</th> <th>Setting value</th> </tr> </thead> <tbody> <tr> <td>1</td> <td>Inc.</td> <td>Incremental position detection system</td> <td>Other than 1,2,3,4,</td> </tr> <tr> <td>2</td> <td>With Stopper</td> <td rowspan="3">Dogless-type absolute position detection system</td> <td>Machine end stopper method</td> </tr> <tr> <td>3</td> <td>No Stopper</td> <td>Marked point alignment method</td> </tr> <tr> <td>4</td> <td></td> <td>Marked point alignment method II</td> </tr> <tr> <td>5</td> <td>Dog-type</td> <td>Dog-type absolute position detection system</td> <td>3</td> </tr> </tbody> </table>		Displayed expression	Type of zero point initialization for absolute position	Setting value	1	Inc.	Incremental position detection system	Other than 1,2,3,4,	2	With Stopper	Dogless-type absolute position detection system	Machine end stopper method	3	No Stopper	Marked point alignment method	4		Marked point alignment method II	5	Dog-type	Dog-type absolute position detection system	3
	Displayed expression	Type of zero point initialization for absolute position	Setting value																				
1	Inc.	Incremental position detection system	Other than 1,2,3,4,																				
2	With Stopper	Dogless-type absolute position detection system	Machine end stopper method																				
3	No Stopper		Marked point alignment method																				
4			Marked point alignment method II																				
5	Dog-type	Dog-type absolute position detection system	3																				
(3) Progress state	<table border="1"> <tr> <td>State</td> <td>This displays the progress state of zero point initialisation. (Note)</td> </tr> <tr> <td>Machine end</td> <td>This displays the distance from the mechanical basic position to the first grid point.</td> </tr> <tr> <td>Machine posn</td> <td>This displays the current machine position. "Not passed" will appear until the axis passes a grid point for the first time.</td> </tr> </table>	State	This displays the progress state of zero point initialisation. (Note)	Machine end	This displays the distance from the mechanical basic position to the first grid point.	Machine posn	This displays the current machine position. "Not passed" will appear until the axis passes a grid point for the first time.																
State	This displays the progress state of zero point initialisation. (Note)																						
Machine end	This displays the distance from the mechanical basic position to the first grid point.																						
Machine posn	This displays the current machine position. "Not passed" will appear until the axis passes a grid point for the first time.																						
(4) Data No.	<p>This displays the No., name, and data used to establish the absolute position.</p> <ul style="list-style-type: none"> • 0 to 2 : Internal data • 2050 to 2059 : Absolute position parameters <p>These can be set also in Absolute position parameter screen. Refer to "Setup Manual" for explanation and setting range of the parameters #2050 to #2059.</p>																						
(5) Name																							
(6) Data																							

Display items	Details
(7) Procedures	This shows the procedures to establish the coordinate system by zero point initialization. The contents differ depending on the type of zero point initialisation.
(8) Initialization message	This displays a message with the background yellow if an illegal value has been set for the zero point initialization. Refer to the section "Appendix 6. Alarms" for details.

(Note) [Type of zero point initialization] indicates the progress state of zero point initialization as shown below.

1. Dogless-type absolute position detection system Machine end stopper method (manual initialization)		
Order	Display	Details of progress
1	Illegality	While the absolute position is lost.
2	Stopper	From when the menu "Abs pos set" is pressed to when the axis pushes against the machine end stopper.
3	Release	From when the axis pushes against the machine end stopper to when axis starts to move in the counter direction.
4	Ret. Ref. P.	From when the axis moves in the counter direction to when it reaches the grid point immediately before the stopper.
5	Complete	The axis has reached the grid point immediately before the stopper. The absolute position has been established.
2. Dogless-type absolute position detection system Machine end stopper method (automatic initialization)		
Order	Display	Details of progress
1	Illegality	While the absolute position is lost.
2	Jog Start	From when the menu "Abs pos set" is pressed to when a value is set in "Zero-P".
3	Stopper1	From when JOG operation starts to when the axis pushes against the machine end stopper.
4	Zero-P. Rel.	From when the axis pushes against the machine end stopper to when axis reaches the approach point.
5	Stopper2	From when the axis reaches the grid point immediately before the stopper to when it pushes against the machine end stopper.
6	Ret. Ref. P.	From when the axis pushes against the machine end stopper second time to when axis reaches the grid point immediately before the stopper.
7	Complete	The axis has reached the grid point immediately before the stopper. The absolute position has been established.
3. Dogless-type absolute position detection system Marked point alignment method		
Order	Display	Details of progress
1	Illegality	While the absolute position is lost.
2	Origin Set	From when the menu "Abs pos set" is pressed to when "1" is set in "Origin-P".
3	Ret. Ref. P.	While moving in the direction designated with the absolute position parameter #2050.
4	Complete	The axis has reached the grid point immediately before the marked point. The absolute position has been established.
4. Dogless-type absolute position detection system Marked point alignment method II		
Order	Display	Details of progress
1	Illegality	While the absolute position is lost.
2	Origin Set	From when the menu "Abs pos set" is pressed to when "1" is set in "Origin-P".
3	Complete	With the sequence above, the absolute position is established.
5. Dog-type absolute position detection system		
Order	Display	Details of progress
1	Illegality	While the absolute position is lost.
2	Zero-P. Rel.	After dog-type manual or automatic reference position return is started.
3	Complete	Axis has returned to the reference position.

Menus

Menu	Details	Type	Reference
Axis select	After selecting this menu, set the axis name, and press the <input type="text" value="INPUT"/> key to display the axis name and related data.	A	6.6.1 Selecting an Axis
Prev axis	This switches the data display to the previous axis' data When the first data is displayed, the last axis data will be displayed.	A	
Next axis	This switches the data display to the next axis' data When the last data is displayed, the first axis data will be displayed.	C	

6.6.1 Selecting the Axis

Procedures

- (1) → The menu is highlighted.
The cursor appears in the input area.
- (2) Set the axis name (which was set with "#1022 axname2") and press key. → The axis name and data change.
Procedures displays according to the selected initialization type.

6.6.2 Carrying Out Dogless-type Zero Point Initialization

Procedures (Machine end stopper method : manual method)

Set the following parameter beforehand in the "Abs.Posit.Param." screen.

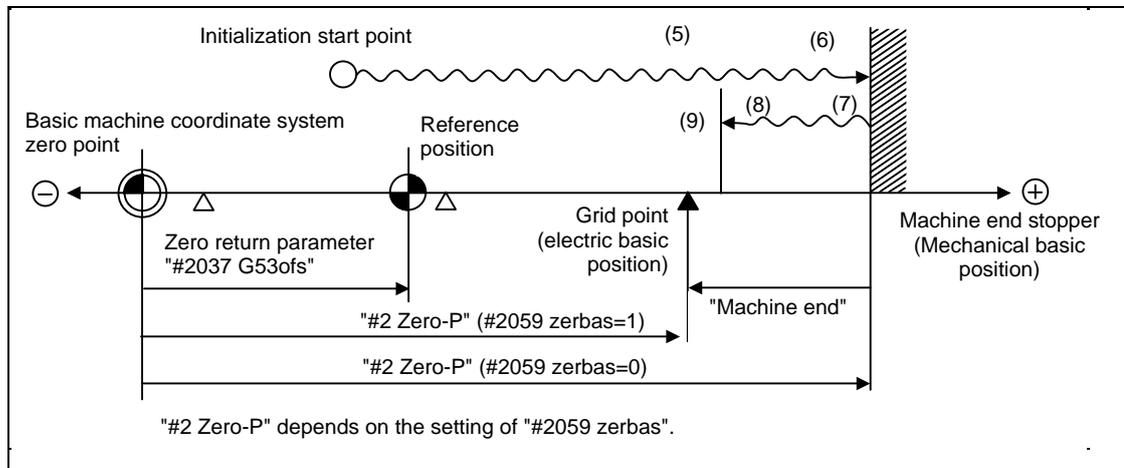
Parameter	Setting value
#2049 type (absolute position detection type)	1 (machine end stopper method is selected)
#2054 clpush (current limit)	0 to 100
#2059 zerbas (zero point return parameter)	1 (on the grid point immediately before the stopper)

The parameters other than "#2049 type" can be set in the "Zero point set" screen.

- (1) Select the axis that "#2049 type" is "1". → Refer to "6.6.1 Selecting the Axis" for procedures to select the axis.
- (2) Select the HANDLE mode or JOG mode.
- (3) Input "1" to "Absolute posn set". → The input value appears at "Absolute posn set".
- (4) Input an arbitrary value to "#2 Zero-P". → The input value appears at "#2 Zero-P".
- (5) Move the axis toward the machine end stopper.
-> a → State : [Stopper]
Machine end : Distance between stopper and grid point
Machine posn : [Not passed] -> current machine position
- (6) Hit against the machine end stopper.
-> b → State : [Stopper]
- The current has reached its limit for given time.
-> c → State : [Release]
- (7) Move the axis in the counter direction.
-> d → State : [Ret. Ref. P.]
- The axis has reached the grid point immediately before the stopper.
-> e → State : [Complete]

This completes zero point initialization. Turn the power OFF and ON after the initialization for all axes.

If "#2059 zerbas" is "0" (absolute position origin point = mechanical basic position), axis will stop automatically at this point without moving in the opposite direction and "Complete" appears at [State] and "0.000" at [Machine end].



- (Note1)** To change just the basic machine coordinate zero point, set "#0 Absolute posn set" and "#2 Zero-P", and then turn the power OFF and ON.
- (Note2)** If pressing against the machine end is attempted without passing the grip point once after turning the power ON, the operation message "Not passed on grid" will appear. Return to a point before the last grid, and then repeat from step of pressing against the machine end stopper.
- (Note3)** If the first grid point is covered by the grid mask (#2028 grmask) as a result of return to the electric basic position, the axis stops at the next grid point.
Note that zero-point shift amount (#2027 G28sft) is invalid.

Procedures (Machine end stopper method : automatic method)

Set the following parameter beforehand in the "Abs.Posit.Param." screen.

Parameter	Setting value
#2049 type (absolute position detection type)	1 (machine end stopper method is selected)
#2054 clpush (current limit)	0 to 100
#2055 pushf (push speed)	1 to 999
#2056 aproch (approach point)	0 to 999.999
#2059 zerbas (zero point return parameter)	1 (on the grid point immediately before the stopper)

The parameters other than "#2049 type" can be set in the "Zero point set" screen.

- (1) Select the axis that "#2049 type" is "1". ➔ Refer to "6.6.1 Selecting the Axis" for procedures to select the axis.

 - (2) Select the "Auto init set" mode.

 - (3) Input "1" to "Absolute posn set". ➔ The input value appears at "Absolute posn set".

 - (4) Input an arbitrary value to "#2 Zero-P". ➔ The input value appears at "#2 Zero-P".

 - (5) Start JOG operation. ➔ State : [Stopper1]
Machine posn : [Not passed] -> current machine position

 - The axis moves toward the machine end stopper at the push speed (#2055).
-> a ➔ State : [Stopper1]

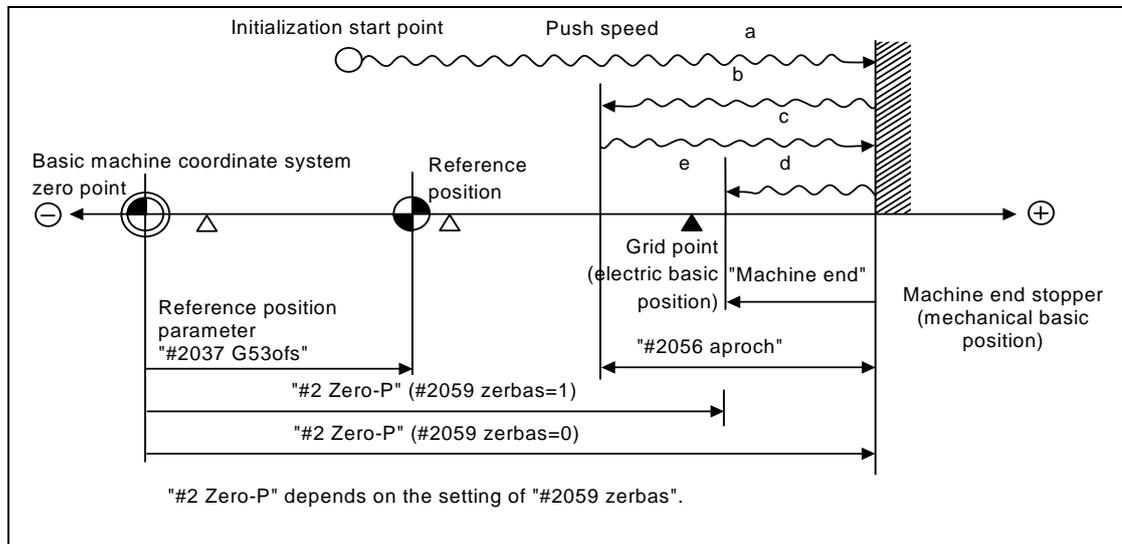
 - The axis pushes against the machine end stopper. After the current reached its limit for given time, the axis returns toward the approach point at the "push speed".
-> b ➔ State : [Zero-P. Rel.]

 - After arrived at the approach point, the axis moves toward the machine end stopper at the "push speed".
-> c ➔ State : [Stopper2]

 - The axis pushes against the machine end stopper. After the current reached its limit for given time, the axis returns toward the grid point immediately before the stopper at the "push speed".
-> d ➔ State : [Ret. Ref. P.]

 - The axis has reached the grid point immediately before the stopper.
-> e ➔ State : [Complete]
Machine end : Distance between stopper and grid point
- This completes zero point initialization. Turn the power OFF and ON after the initialization for all axes.

If "#2059 zervas" is "0" (absolute position origin point = mechanical basic position), axis will stop automatically at this point without moving in the opposite direction and "Complete" appears at [State] and "0.000" at [Machine end].

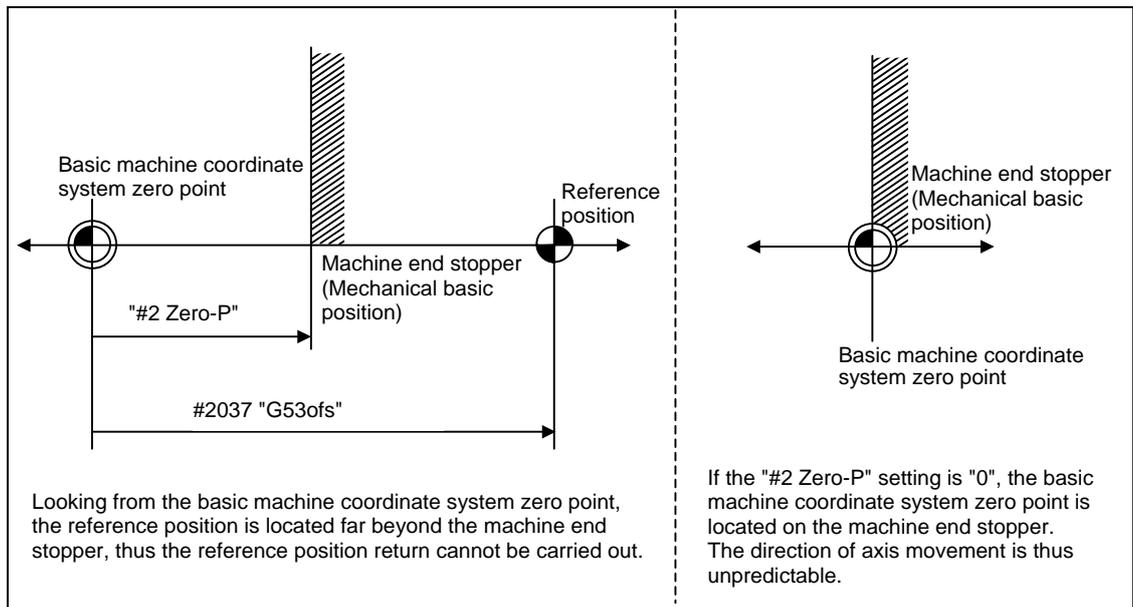


- (Note1)** To change just the basic machine coordinate zero point, set "#0 Absolute posn set" and "#2 Zero-P", and then turn the power OFF and ON.
- (Note2)** If pressing against the machine end is attempted without passing the grip point once after turning the power ON, the operation message "Not passed on grid" will appear. Return to a point before the last grid, and then repeat from step of pressing against the machine end stopper.
- (Note3)** If the first grid point is covered by the grid mask (#2028 grmask) as a result of return to the electric basic position, the axis stops at the next grid point.
Note that zero-point shift amount (#2027 G28sft) is invalid.
- (Note4)** Acceleration/deceleration during movement at the specified push speed is performed in smoothing-off (step feed) mode.
- (Note5)** If "0" is specified for "#2056 aproch" of the absolute position parameters, the machine zero point is regarded as the approach point.
- (Note6)** Automatic initialization is interrupted if one of the following events occurs. If it is interrupted, [State] indicates "Jog Start" (after selecting the "Auto init set" mode if it is caused by mode change), so restart operation from the step of JOG-start.
- An absolute position detection alarm occurs.
 - Operation preparation signal turns OFF.
 - The mode is changed.
 - The system is reset.
- If [State] is "Complete" before automatic initialization is started, "State" returns to "Complete" when power is turned OFF and ON again without restarting the operation.

(Note7) Automatic initialization cannot be started in the following cases. The operation message "Can't start" will appear if starting is attempted.

- When "#0 Absolute posn set" is not set.
- When the "#2 Zero-P" setting is inappropriate.
- When "#2055 pushf" is not set.
- When "Z71 Abs encoder failure 0005" has occurred.

In the above cases, if the "#2 Zero-P" setting is inappropriate, this means that the relation of "#2 Zero-P" and "#2037 G53ofs" is inappropriate. That is, if "#2 Zero-P" is smaller than the "#2037 G53ofs", the machine end stopper will be located between the basic machine coordinate system zero point and reference position; this disables automatic initialization. (Refer to the following left figure.) If "#2 Zero-P" is set to "0", the machine end stopper direction is unpredictable; this also disables automatic initialization. (Refer to the following right figure.)



Procedures (Marked point alignment method)

Set the following parameter beforehand in the "Abs.Posit.Param." screen.

Parameter	Setting value
#2049 type (absolute position detection type)	2 (marked point alignment method is selected)
#2050 absdir (basic Z - direction)	0/1

- (1) Select the axis that "#2049 type" is "2". ➔ Refer to "6.6.1 Selecting the Axis" for procedures to select the axis.

 - (2) Select the HANDLE mode, HANDLE-AX mode, or JOG mode.

 - (3) Input "1" to "#0 Absolute posn set". ➔ The input value appears at "#0 Absolute posn set".

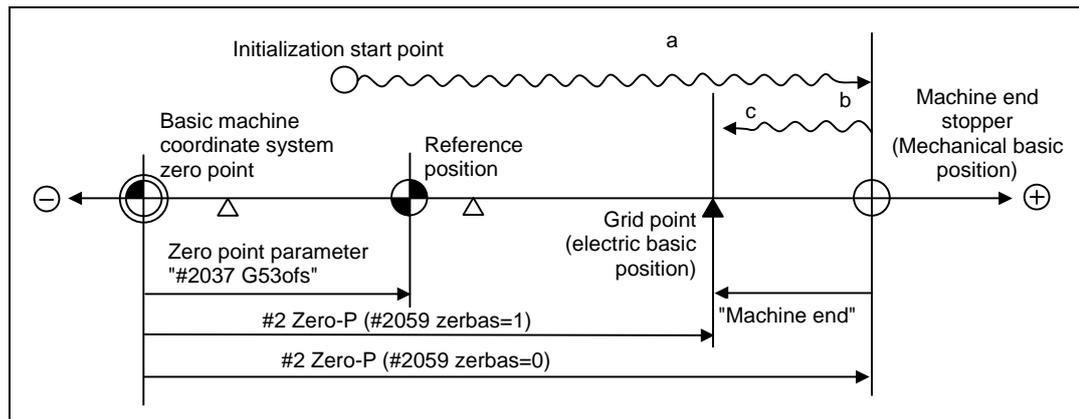
 - (4) Input an arbitrary value to "#2 Zero-P". ➔ The input value appears at "#2 Zero-P".

 - (5) Move the axis toward the mechanical basic position and align it to the marked point.
-> a ➔ State : [Origin Set]
Machine posn : [Not passed] -> current machine position

 - (6) Input "1" to "#1 Origin-P". ➔ The input value appears at "#1 Origin-P".

 - (7) Move the axis in the direction designated with the parameter "#2050 absdir".
-> b ➔ State : [Ret. Ref. P.]
Machine end : Distance between machine basic point (marked point) and the first grid point

 - The axis reaches the first grid point.
-> c ➔ State : [Complete]
- This completes zero point initialization. Turn the power OFF and ON after the initialization for all axes.



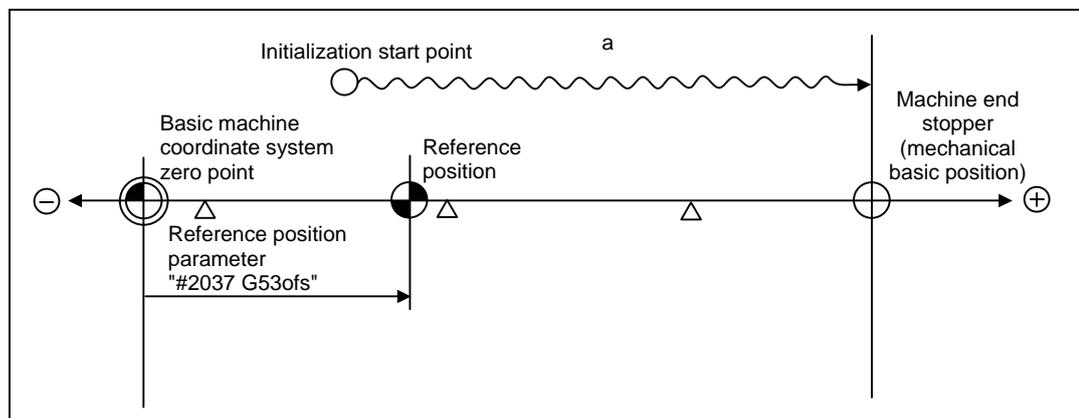
- (Note 1)** To change just the basic machine coordinate zero point, set "#0 Absolute posn set" and "#2 Zero-P", and then turn the power OFF and ON.
- (Note 2)** If aligning axis on the marked point is attempted without passing the grip point once after turning the power ON, the operation message "Not passed on grid" will appear. Return to a point before the last grid, and then repeat from step of aligning the axis on the marked point.
- (Note 3)** If the first grid point is covered by the grid mask (#2028 grmask) as a result of return to the electric basic position, the axis stops at the next grid point.
Note that zero-point shift amount (#2027 G28sft) is invalid.
- (Note 4)** Reconfirm the "absdir" setting if the machine does not move in the direction of "#2050 absdir". The machine will move only in the positive direction when set to "0", and the negative direction when set to "1".

Procedures (Marked point alignment method II)

Set the following parameter beforehand in the "Abs.Posit.Param." screen.

Parameter	Setting value
#2049 type (absolute position detection type)	4 (marked point alignment method II is selected)

- (1) Select the axis that "#2049 type" is "4". → Refer to "6.6.1 Selecting the Axis" for procedures to select the axis.
 - (2) Select the HANDLE mode, HANDLE FEED AXIS SELECT mode, or JOG mode.
 - (3) Input "1" to "#0 Absolute posn set". → The input value appears at "#0 Absolute posn set".
 - (4) Input an arbitrary value to "#2 Zero-P". → The input value appears at "#2 Zero-P".
- Move the axis toward the mechanical basic position and align it to the marked point.
-> a
- State : [Origin Set]
- Input "1" to "#1 Origin-P". → The input value appears at "#1 Origin-P".
State : [Complete]
Machine end : 0.000
Machine posn : The value set in "#2 ZERO"



- (Note1)** To change just the basic machine coordinate zero point, set "#0 Absolute posn set" and "#2 Zero-P", and then turn the power OFF and ON.
- (Note2)** If aligning axis on the marked point is attempted without passing the grip point once after turning the power ON, the operation message "Not passed on grid" will appear. Return to a point before the last grid, and then repeat from step of aligning the axis on the marked point.
- (Note3)** If the first grid point is covered by the grid mask (#2028 grmask) as a result of return to the electric basic position, the axis stops at the next grid point.
Note that zero-point shift amount (#2027 G28sft) is invalid.

6.6.3 Carrying Out Dog-type Zero Point Initialization

Procedures (Marked point alignment method II)

Set the following parameter beforehand in the "Abs.Posit.Param." screen.

Parameter	Setting value
#2049 type (absolute position detection type)	3 (dog-type method is selected)

- (1) Select the axis that "#2049 type" is "3". → Refer to "6.6.1 Selecting the Axis" for procedures to select the axis.
- (2) Carry out the manual or automatic dog-type reference position return. →
- | | |
|--------------------------------------|---------------------|
| Reference position return completes. | State : [Complete] |
| | Machine end : 0.000 |
- This completes zero point initialization. Turn the power OFF and ON after the initialization for all axes.

(Note1) If the dog-type reference position return is interrupted by resetting, the previous state ("Complete" or "Illegality") will display in the [State] column.

(Note2) With dog-type reference position return, reference position return can be executed again even if the [State] is "Complete".

6.6.4 Precautions

Precautions common for the initialization operation

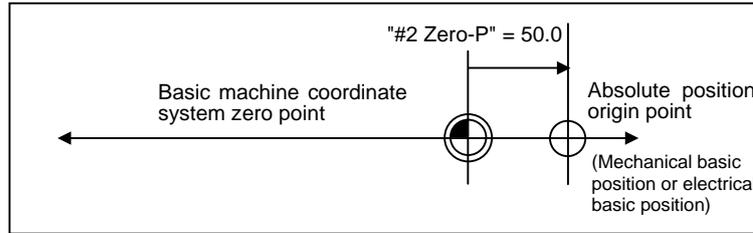
- (1) The "#0 Absolute posn set" parameter (axis for which zero point is to be initialized) can be set simultaneously for all axes or individually for each axis.
- (2) The "#0 Absolute posn set" parameter cannot be turned OFF with the keys. It is turned OFF when the power is turned ON again.
- (3) "#2 ZERO-P" can be set at any time as long as "#0 Absolute posn set" is set to "1".
- (4) The grid point must be passed at least once after turning the power ON before initializing the zero point. If the grid point has not been passed, the operation message "Not passed on grid" will appear at the "Machine posn".
- (5) When the absolute position is established, the required data will be stored in the memory.

Precautions common for the dogless-type absolute position detector

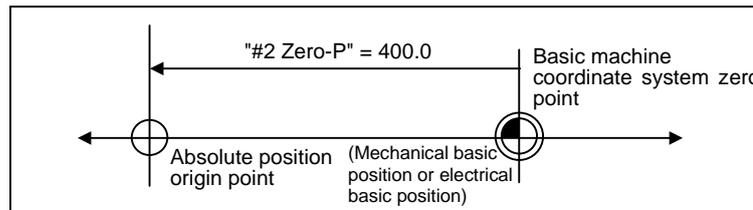
(1) Example of setting "#2 Zero-P" parameter

For the "#2 Zero-P" parameter, set the coordinate value of the absolute position origin point (mechanical basic position or electrical basic position) looking from the basic machine coordinate system zero point.

(Example 1) To set the zero point at 50.0mm before absolute position origin point on + end



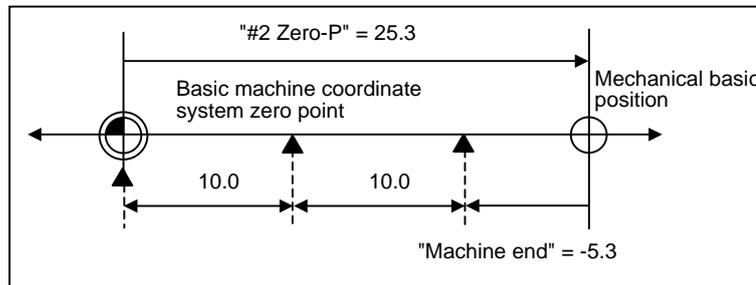
(Example 2) To set the zero point at 400.0mm before the machine basic position or absolute position origin point on - end.



(Example 3) To set the basic machine coordinate system zero point on the grid point, calculate the "#2 Zero-P" parameter setting value as shown below using the value displayed at "Machine end". "Machine end" shows the distance from the mechanical basic position to the previous grid point.

(Note that when setting the electrical basic position coordinate value in "#2 Zero-P", the "Machine end" value does not need to be considered.)

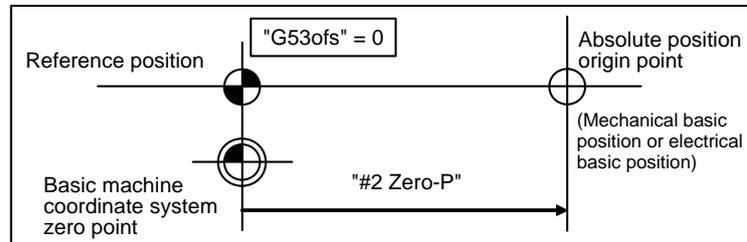
To set the third grid point as the zero point when the "Machine end" display is -5.3 at the + end basic position. (Example of 10.0mm grid interval.)



(2) Setting the reference position

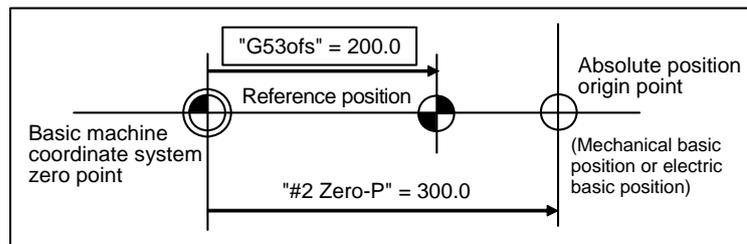
The reference position can be set as shown below by setting the "#2037 G53ofs".

(Example 1) To set the reference position to the same position as the basic machine coordinate system zero point.



(Example 2) To set the reference position at a position 200.0mm to the + side from the basic machine coordinate zero point.

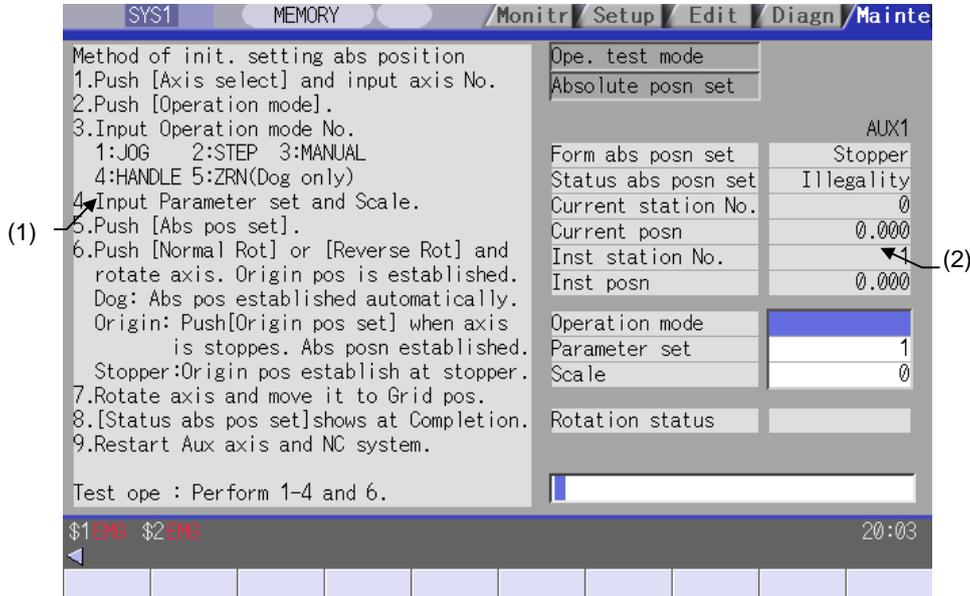
(To set the basic machine coordinate system zero point 300.0mm front of the absolute position origin point.)



6.7 Auxiliary Axis Test Screen

The absolute position setting and test operations can be carried out by separating the auxiliary axis control from the PLC.

For PLC indexing axis, the absolute position setting and test operation can be carried out on this screen. In 70 series, these operations can be carried out for only PLC indexing axis.



Display items

Display item	Details	Display items/setting range
(1) Operation explanation area	The operation method is displayed. Refer to "MR-J2-CT Series SPECIFICATIONS AND INSTRUCTION MANUAL" (BNP-B3944) for details.	
(2) Setting and display area		
Ope. test mode	This is highlighted when the absolute position can be initialized or test operation is possible. This cuts off the signals from the PLC.	
Absolute posn set	This is highlighted when the absolute position can be initialized.	
Form abs posn set	This displays the initialization method of the absolute position set with the auxiliary axis parameter "#50120 ABS Type".	Dog/Stopper/Origin (Note 1)
Status abs posn set	This displays the initialization state of the absolute position. Refer to "Display of "Status abs posn set" " in 6.7.2 for details.	Complete/Illegality/Stopper/Release/Ret. Ref. P./Origin Set/Zero-P. Rel.
Current station No.	This displays the number of the currently stopped station.	This follows the "#50100 station" setting. (Note 2)
Current posn	This displays the current coordinate position.	0 to 360°
Inst station No.	This displays the commanded station No. for automatic operation, or the nearest station No. to be stopped at for manual operation.	This follows the "#50100 station" setting. (Note 2)
Inst posn	This displays the coordinate position corresponding to the target station No.	0 to 360°

(Note 1) For PLC indexing axis, the absolute position detection method set by "#2049 type" is displayed. However, the dog-type method initializing cannot be carried out.

(Note 2) For PLC indexing axis, this follows "#12801 aux_station" setting.

Display items	Details	Display items/setting range
(Continued from the previous page)		
Operation mode	This displays the currently selected operation mode.	1: JOG 2: STEP 3: MANUAL 4: HANDLE 5: ZRN (Dog only)
Parameter set	This displays the No. of the currently selected operation parameter group.	1: Parameter set 1 2: Parameter set 2 3: Parameter set 3 4: Parameter set 4
Scale	This displays the feed amount scale. Feed amount = 0.001 degree * Scale	0: 1-fold 1: 10-fold 2: 100-fold 3: 1000-fold
Rotation status	This displays the state of the auxiliary axis or PLC indexing axis.	Normal Rot/Reverse Rot/Stop

Menus

Menu	Details	Type	Reference
Axis select	This designates the auxiliary axis No. for which initialization or test operation is to be carried out. After pressing this menu, set the auxiliary axis No. (1 to 6).	A	
OpeTest mode	This switches ON/OFF for the operation adjustment mode. When absolute position initialization set and test operation are carried out, select the operation adjustment mode. (Note) The operation adjustment mode cannot be selected in the following cases. <For auxiliary axis> J2CT operation adjustment mode valid signal (R9998/bit0) is OFF. The auxiliary axis is being executed. <For PLC indexing axis> J2CT operation adjustment mode valid signal (R9998/bit0) is OFF. The auxiliary axis is being executed.	C	
Abs pos set	This turns ON absolute position initialization set mode. This turns OFF absolute position initialization set mode after the absolute position initialization set is completed. This menu can be selected only in the operation adjustment mode.	C	
Origin pos set	When the display item "Form abs posn set" is set to "Origin", press this menu at the position to be used as the origin point. The absolute position will then be set at this position.	C	
Normal Rot	The axis is started with forward run. (1) For HANDLE A forward run cannot be started by pressing this menu. (2) For MANUAL/JOG operation When this menu is pressed in forward run, the axis stops. When this menu is pressed in reverse run, the axis is started with forward run after it stops at once. (3) For STEP Whenever this menu is pressed, forward run is executed by each feed amount. This menu can be selected in operation adjustment mode only.	B	
Reverse Rot	The axis is started with reverse run. (1) For HANDLE A reverse run cannot be started by pressing this menu. (2) For MANUAL/JOG operation When this menu is pressed in reverse run, the axis stops. When this menu is pressed in forward run, the axis is started with reverse run after it stops at once. (3) For STEP Whenever this menu is pressed, reverse run is executed by each feed amount. This menu can be selected in operation adjustment mode only.	B	

6.7.1 Preparation

Before the screen is switched to auxiliary axis test screen, prepare the following things.

For auxiliary axis

- (1) Parameter settings
Set the following parameters.

No.	Name	Details																						
50102	Cont2 Control parameter 2	bit7 = 1 (Absolute position detection) Turn the NC power ON again after this parameter is set. (Note) The alarm "Z70 Abs data error" occurs after the power ON again.																						
50120	ABS Type Absolute position detection parameter	<p>bit1/bit2 : Select the absolute position detection method.</p> <table border="1"> <thead> <tr> <th>Method</th> <th>bit2</th> <th>bit1</th> <th>Detail</th> </tr> </thead> <tbody> <tr> <td>Dog-type method</td> <td>(Note)</td> <td>1</td> <td>This decides the reference position by the near point dog.</td> </tr> <tr> <td>Mechanical end stopper method</td> <td>0</td> <td>0</td> <td>This decides the basic point by pushing an axis to machine end etc. when the torque (current) limit is set.</td> </tr> <tr> <td>Marked point alignment method</td> <td>1</td> <td>0</td> <td>This decides the basic point by aligning an axis to machine origin point.</td> </tr> </tbody> </table> <p>(Note) When bit1 is "1", dog-type method is selected regardless of bit2 setting.</p> <p>bit3 : Select the electrical basic position direction at marked point alignment method.</p> <table border="1"> <thead> <tr> <th>Direction</th> <th>bit3</th> </tr> </thead> <tbody> <tr> <td>Electrical basic position direction +</td> <td>0</td> </tr> <tr> <td>Electrical basic position direction -</td> <td>1</td> </tr> </tbody> </table>	Method	bit2	bit1	Detail	Dog-type method	(Note)	1	This decides the reference position by the near point dog.	Mechanical end stopper method	0	0	This decides the basic point by pushing an axis to machine end etc. when the torque (current) limit is set.	Marked point alignment method	1	0	This decides the basic point by aligning an axis to machine origin point.	Direction	bit3	Electrical basic position direction +	0	Electrical basic position direction -	1
Method	bit2	bit1	Detail																					
Dog-type method	(Note)	1	This decides the reference position by the near point dog.																					
Mechanical end stopper method	0	0	This decides the basic point by pushing an axis to machine end etc. when the torque (current) limit is set.																					
Marked point alignment method	1	0	This decides the basic point by aligning an axis to machine origin point.																					
Direction	bit3																							
Electrical basic position direction +	0																							
Electrical basic position direction -	1																							

- (2) Release servo OFF/interlock for auxiliary axis.
- (3) Turn J2CT operation adjustment mode valid signal (R9998/bit0) ON.

For PLC indexing axis

(1) Parameter settings

Set the following parameters.

No.	Name	Details
2049	type Absolute position detection method	Select the absolute position zero point alignment method. 0: Not absolute position detection 1: Stopper method (push against mechanical stopper) 2: Marked point alignment method (align to alignment point) 3: Dog-type (align with dog and near point detection switch) 4: Marked point alignment method II (Align to alignment mark. Grid return won't be performed after marked point alignment) 9: Simple absolute position (Not absolute position detection, but the position when the power is turned off is registered.) After this parameter is set, turn ON the NC power supply again.

(2) Release servo OFF/interlock for PLC indexing axis.

(3) Turn J2CT operation adjustment mode valid signal (R9998/bit0) ON.

6.7.2 Selecting a Device, Directory and File

Operation method (dog type)

- (1) Press the menu , input the axis No.  If the axis No. that does not exist is designated, an error will occur.
- (2) Press the menu .  When the J2CT operation adjustment mode valid signal from PLC is only turning ON while an auxiliary axis is stopping, the menu can be turned ON.
- (3) Operation mode is changed to "Zero-P. Rel."
- (4) Set the "Parameter set" No.  If the value other than 1 to 4 is set to "Parameter set", a setting error will occur.
- (5) Press the menu or , start the axis.  The absolute position is set automatically.
- (6) Power ON again.

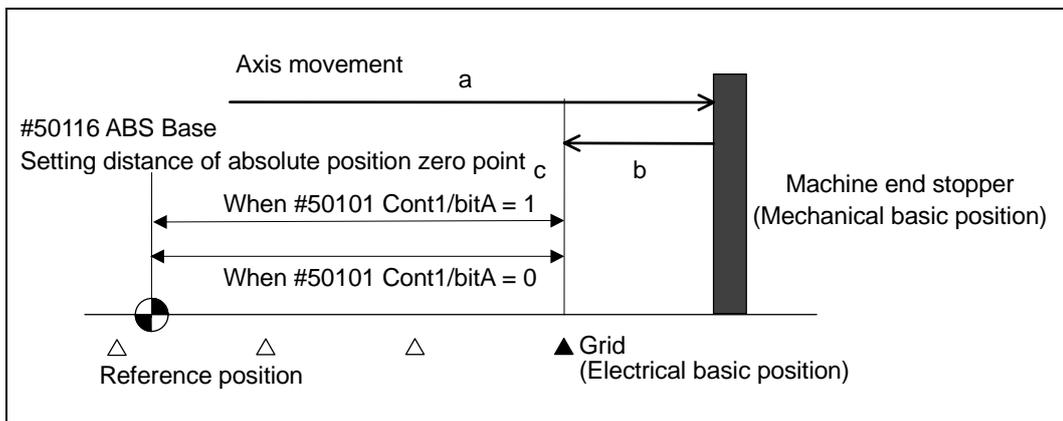
(Note) For PLC indexing axis, the dog-type reference position return cannot be carried out.

Operation methods (Stopper method) [For auxiliary axis]

- | | | | |
|-----|---|---|---|
| (1) | Press the menu Axis select , input the axis No. | ➔ | If the axis No. that does not exist is designated, an error will occur. |
|-----|---|---|---|
- | | | | |
|-----|--|---|--|
| (2) | Press the menu OpeTestmode . | ➔ | When the J2CT operation adjustment mode valid signal from PLC is only turning ON while an auxiliary axis is stopping, the menu OpeTestmode can be turned ON. |
|-----|--|---|--|
- | | | | |
|-----|--|---|---------------------------------|
| (3) | Press the menu Abs pos set . | ➔ | Status abs posn set : "Stopper" |
|-----|--|---|---------------------------------|
- | | | | |
|-----|---|--|--|
| (4) | Operation mode is changed to "HANDLE" or "JOG". | | |
|-----|---|--|--|
- | | | | |
|-----|-----------------------------|---|---|
| (5) | Set the "Parameter set" No. | ➔ | If the value other than 1 to 4 is set to "Parameter set", a setting error will occur. |
|-----|-----------------------------|---|---|
- | | | | |
|-----|--|---|---|
| (6) | When the operation mode is HANDLE, set the feed amount per pulse to "Scale". | ➔ | If the value other than 0 to 3 is set to "Scale", a setting error will occur. |
|-----|--|---|---|
- | | | | |
|-----|---|---|---|
| (7) | The axis is traveled to machine end stopper.
JOG : Press the menu Normal Rot
or Reverse Rot
HANDLE: Turn the handle.
-> a | ➔ | Status abs posn set : " Stopper" -> "Release" |
|-----|---|---|---|
- | | | | |
|-----|--|--|--|
| (8) | The axis travels in the opposite direction for stopper direction.
-> b

Reach the first grid.
-> c | | |
|-----|--|--|--|
- | | | | |
|-----|-----------------|--|--|
| (9) | Power ON again. | | |
|-----|-----------------|--|--|

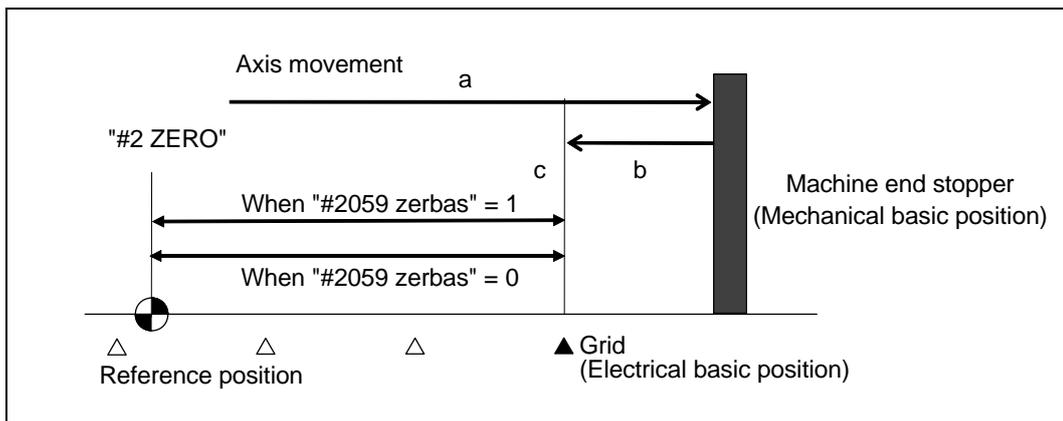
When "#50101 Cont1/bitA" is "0" (absolute position basic point = mechanical basic position), the axis will automatically stop without traveling. When absolute position coordinate zero point is set to mechanical basic position or other than electrical basic position, the reference position is moved with "#50116 ABS Base (absolute position zero point)".



Operation methods (Stopper method) [For PLC indexing axis]

- (1) Press the menu ➔ If the axis No. that does not exist is designated, an error will occur.
- (2) Press the menu ➔ When the J2CT operation adjustment mode valid signal from PLC is only turning ON while a PLC indexing axis is stopping, the menu
- (3) Press the menu ➔ Status abs posn set : "Stopper"
- (4) Operation mode is changed to "HANDLE" or "JOG".
- (5) Set the "Parameter set" No. ➔ If the value other than 1 to 4 is set to "Parameter set", a setting error will occur.
- (6) When the operation mode is HANDLE, set the feed amount per pulse to "Scale". ➔ If the value other than 0 to 3 is set to "Scale", a setting error will occur.
- (7) The axis is traveled to machine end stopper.
 JOG : Press the menu
 HANDLE: Turn the handle.
 -> a ➔ Status abs posn set : " Stopper" -> "Release"
- (8) The axis travels in the opposite direction for stopper direction.
 -> b
- > c
- (9) Power ON again.

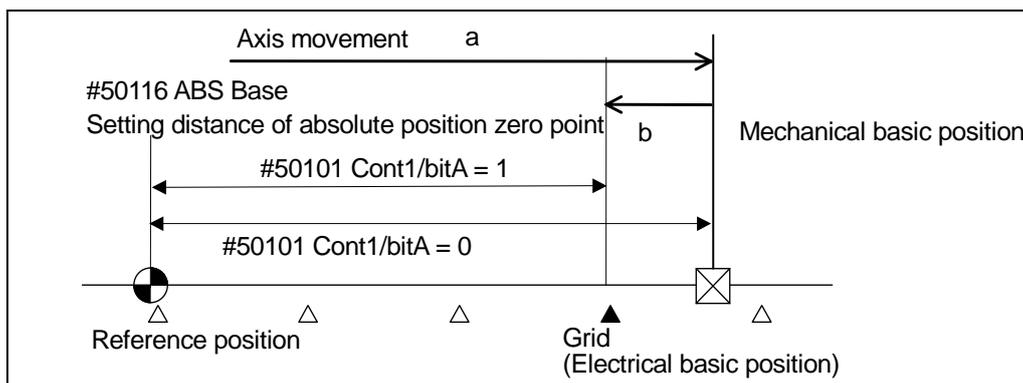
When "#2059 zerbass" is "0" (absolute position basic point = mechanical basic position), the axis will automatically stop without traveling. When absolute position coordinate zero point is set to mechanical basic position or other than electrical basic position, the reference position is moved by setting the coordinate position of basic machine coordinate to "ZERO" on the absolute position setting screen. (Refer to "6.6 Absolute Position Setting Screen" for operation method.)



Operation methods (Marked point alignment method) [For auxiliary axis]

- (1) Press the menu ➔ If the axis No. that does not exist is designated, an error will occur.
- (2) Press the menu ➔ When the J2CT operation adjustment mode valid signal from PLC is only turning ON while an auxiliary axis is stopping, the menu
- (3) Press the menu ➔ Status abs posn set : "Origin Set"
- (4) Operation mode is changed to "HANDLE", "STEP" or "JOG".
- (5) Set the "Parameter set" No. ➔ If the value other than 1 to 4 is set to "Parameter set", a setting error will occur.
- (6) When the operation mode is STEP or HANDLE, set the feed amount per pulse to "Scale". ➔ If the value other than 0 to 3 is set to "Scale", a setting error will occur.
- (7) Press the menu ➔ Status abs posn set : "Ret. Ref. P."
- (8) The axis is traveled to mechanical basic position, aligned origin point.
 JOG/STEP: Press the menu
 HANDLE: Turn the handle.
 -> a
- ➔ Status abs posn set : "Complete"
 -> c
- (9) Power ON again.

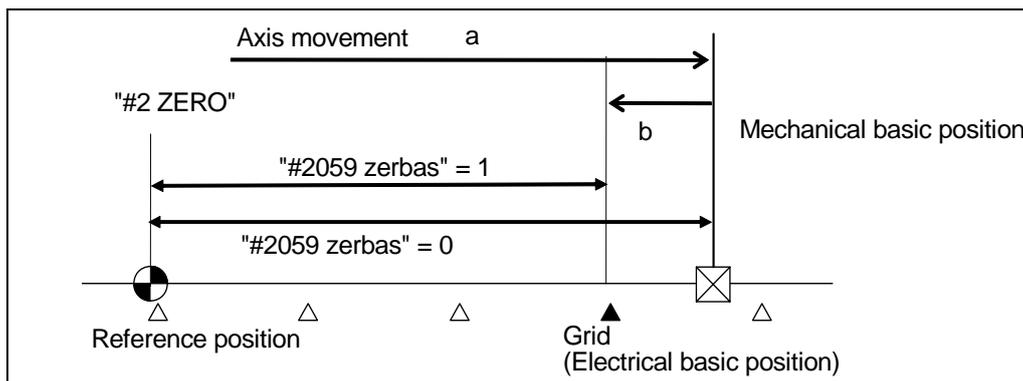
When "#50101 Cont1/bitA" is "0" (absolute position basic point = mechanical basic position), the axis will automatically stop without traveling. When absolute position coordinate zero point is set to mechanical basic position or other than electrical basic position, the reference position is moved with "#50116 ABS Base (absolute position zero point)".



Operation methods (Marked point alignment method) [For PLC indexing axis]

- (1) Press the menu ➔ If the axis No. that does not exist is designated, an error will occur.
- (2) Press the menu ➔ When the J2CT operation adjustment mode valid signal from PLC is only turning ON while a PLC indexing axis is stopping, the menu
- (3) Press the menu ➔ Status abs posn set : "Origin Set"
- (4) Operation mode is changed to "HANDLE", "STEP" or "JOG".
- (5) Set the "Parameter set" No. ➔ If the value other than 1 to 4 is set to "Parameter set", a setting error will occur.
- (6) When the operation mode is STEP or HANDLE, set the feed amount per pulse to "Scale". ➔ If the value other than 0 to 3 is set to "Scale", a setting error will occur.
- (7) Press the menu ➔ Status abs posn set : "Ret. Ref. P."
- (8) The axis is traveled to mechanical basic position, aligned origin point.
 JOG/STEP: Press the menu
 HANDLE: Turn the handle.
 -> a
- ➔ Status abs posn set : "Complete"
 -> c
- (9) Power ON again.

When "#2059 zerbas" is "0" (absolute position basic point = mechanical basic position), the axis will automatically stop without traveling. When absolute position coordinate zero point is set to mechanical basic position or other than electrical basic position, the reference position is moved by setting the coordinate position of basic machine coordinate to "ZERO" on the absolute position setting screen. (Refer to "6.6 Absolute Position Setting Screen" for operation method.)



Display of "Status abs posn set"

Depending on the state during operation, the display will change as shown below. Refer to the following table for details on each display item.

State/operation	Display of "Status abs posn set"		
	Dog	Stopper	Origin
Absolute position lost	Illegality	Illegality	Illegality
Absolute posn set ON	↓	↓	↓
Start Normal Rot or Reverse Rot	↓	Stopper	Origin Set
Zero-P. Rel.	Zero-P. Rel.	↓	↓
Stopper	↓	Release	↓
Stop Normal Rot or Reverse Rot	↓	↓	↓
Origin pos set ON	↓	↓	Ret. Ref. P.
Start Normal Rot or Reverse Rot	↓	↓	↓
Absolute position established	Complete	Complete	Complete

With operation ———→
 Without operation - - - - -→

Display item of "Status abs posn set"	Details
Illegality	This appears when the absolute position has been lost.
Stopper	This appears after the absolute position initialization set is started and the machine end is pushed against.
Release	This appears after the machine end is pushed against, and the current limit is reached continuously for a set time.
Origin Set	This appears after the absolute position initialization set is started, and the origin point is designated moving to the mechanical basic position.
Ret. Ref. P.	This appears after the basic point is designated, and the nearest grid is reached with the "#50120 ABS TYPE" basic point direction movement. (Note 1)
Zero-P. Rel.	This appears when the reference position has been returned to with the dog-type initialization.
Complete	This appears when the absolute position initialization set has been completed. This displays the state that the absolute position has been established.

(Note 1) For PLC indexing axis, the axis is moved in the direction set with "#2050 absdir".

(Note 2) When using position setting methods other than the dog-type method, mechanical end stopper method, marked point alignment method, the initialization state will be blank.

6.7.3 Test Operation

Operation methods

The test operation can be carried out with menu **Normal Rot/Reverse Rot** by separating the auxiliary axis control from the PLC.

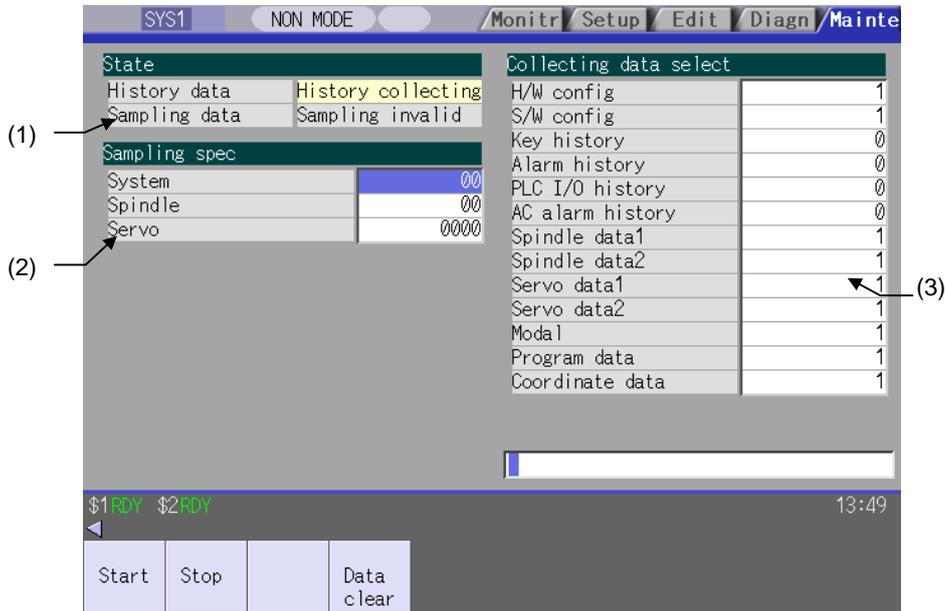
- | | |
|---|--|
| (1) Press the menu Axis select , input the axis No. | ➔ If the axis No. that does not exist is designated, an error will occur. |
| (2) Press the menu OpeTestmode . | ➔ When the J2CT operation adjustment mode valid signal from PLC is only turning ON while an auxiliary axis or PLC indexing axis is stopping, the menu OpeTestmode can be turned ON. |
| (3) Select the "Operation mode". | ➔ If the value other than 1 to 5 is set to "Operation mode", a setting error will occur. |
| (4) When the operation mode is STEP or HANDLE, set the feed amount per pulse to "Scale". | ➔ If the value other than 0 to 3 is set to "Scale", a setting error will occur. |
| (5) Set the "Parameter set" No. | ➔ If the value other than 1 to 4 is set to "Parameter set", a setting error will occur. |
| (6) Rotate the auxiliary axis. (Start by pressing the menu Normal Rot or Reverse Rot .) | <p><JOG>
Rotation will start in the commanded direction. When the same menu is pressed again, the axis will immediately decelerate and stop. If the Normal Rot menu is pressed during reverse run, the rotation will stop, and then forward run will start.</p> <p><MANUAL>
Rotation will start in the commanded direction. When the same menu is pressed again, the axis will be positioned to the nearest station, and will stop. If the Reverse Rot menu is pressed during forward run, the rotation will stop, and then reverse run will start. Note that the rotation cannot be started unless the absolute position is established.</p> <p><INCREMENTAL>
Each time the movement is started, the axis will rotate by the amount set in the Scale. Other operations are disabled during movement.</p> <p><HANDLE>
The axis is fed with handle rotation.</p> |

6.7.4 Precaution

- (1) Do not turn "Auto operation "start" command signal (ST) " ON during Ope. test mode.
- (2) If the screen shifts other screen, the auxiliary axis mode (Ope. test mode, Absolute posn set) is held. However, the axis stops if the axis is rotating.

6.8 Diagnosis Data Collection Setting Screen

The NC automatically collects diagnosis data based on the "Sampling spec" and "Collecting data select" settings specified at this screen.

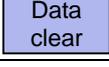


Display items

Display item	Details
(1) State	Display the history data and sampling data collection status.
History data	Display the history data collection status. History collecting : History data collection is in progress. History stop : History data collection is stopped.
Sampling data	Display the sampling data collection status. Sampling : Sampling data collection is in progress. Sampling stop : Sampling data collection is stopped. Sampling invalid : Sampling data collection is disabled.
(2) Sampling spec	Specify the part system, spindle, and servo axes where sampling data is to be collected.
System	Specify the part systems where sampling data is not to be collected. When "0" is set, sampling is carried out for all part systems. This setting is ignored if a non-existent part system is specified.
Spindle	Specify the spindle where sampling data is not to be collected. When "0" is set, sampling is carried out for all spindles. This setting is ignored if a non-existent spindle is specified.
Servo	Specify the servo axes where sampling data is not to be collected. This setting is not system-specific. When "0" is set, sampling is carried out for all axes. This setting is ignored if a non-existent servo axis is specified.

Display item	Details
(3) Collecting data select	This selects the diagnosis data to be collected. (0 : Collected. 1 : Not collected.)
H/W config	This selects whether or not H/W configuration data is collected.
S/W config	This selects whether or not S/W configuration data is collected.
Key history	This selects whether or not key history data is collected.
Alarm history	This selects whether or not alarm history data is collected.
PLC I/O history	This selects whether or not PLC input/output signal history data is collected.
AC alarm history	This selects whether or not AC input power error history data is collected.
Spindle data 1	This selects whether or not spindle current waveform (speed FB, current FB, current command, position FB, position command, droop) data is collected.
Spindle data 2	This selects whether or not spindle current waveform (driver statuses 1,3,4) data is collected.
Servo data 1	This selects whether or not servo current waveform (speed FB, current FB, current command, position FB, position command, droop) data is collected.
Servo data 2	This selects whether or not servo current waveform (detector position Rn, Pn) data is collected.
Modal	This selects whether or not modal information (information for each running block (block data, G-command modal, workpiece coordinate offset)) data is collected.
Program data	This selects whether or not program data (information for each running block (program coordinate position, manual interruption amount, command termination)) data is collected.
Coordinate data	This selects whether or not coordinate data (information for each running block (relative position, machine FB position, machine command position)) data is collected.

Menus

Menu	Details	Type	Reference
	This begins collecting sampling data and history data.	A	6.8.1 Performing a Data Collection Operation
	This stops the sampling data and history data collection operation.	A	6.8.2 Stopping a Data Collection Operation
	This stops the sampling data and history data collection operation, and clears the collected data.	A	6.8.3 Clearing a Collected Data

6.8.1 Performing a Data Collection Operation

Operation methods

- (1) Press the menu .  An operation message "The collection begin?" appears.
- (2) Press or key.  Data collection begins, and "History collecting" and "Sampling" appears in the status column.

6.8.2 Stopping a Data Collection Operation

Operation methods

- (1) Press the menu .  An operation message "The collection stop?" appears.
- (2) Press or key.  Data collection stops, and "History stop" and "Sampling stop" appears in the status column. To resume data collection, press the menu .

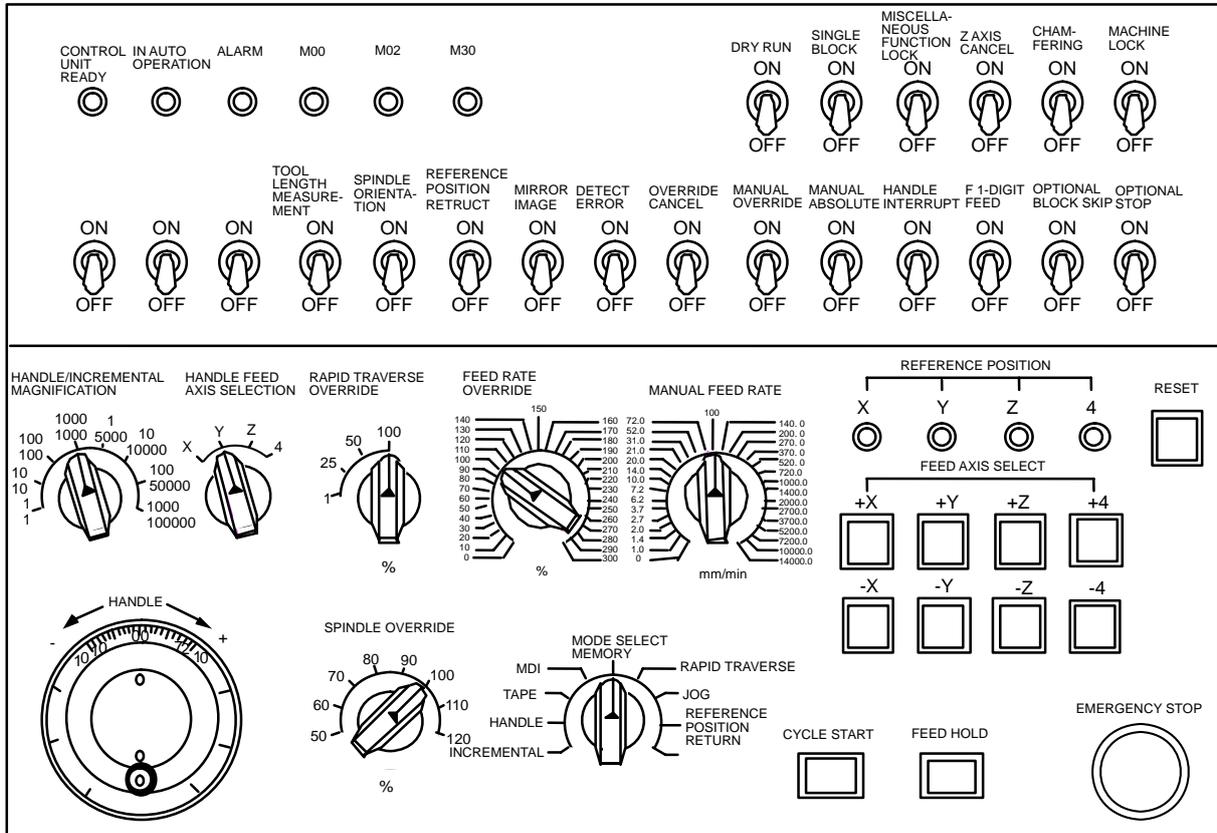
6.8.3 Clearing a Collected Data

Operation methods

- (1) Press the menu .  An operation message "Execute the collection data clear?" appears.
- (2) Press or key.  The collected data is cleared. "Stop" appears in the status column. To resume data collection, press the menu .

II. MACHINE OPERATIONS

This chapter explains the functions and operation method of the machine operation switches for operation (automatic operation and manual operation) by using the illustration of the machine operation panel. The actual machine operation and motion vary by machine specification. Refer to the operation manual issued by the machine tool builder. Use this chapter for reference.

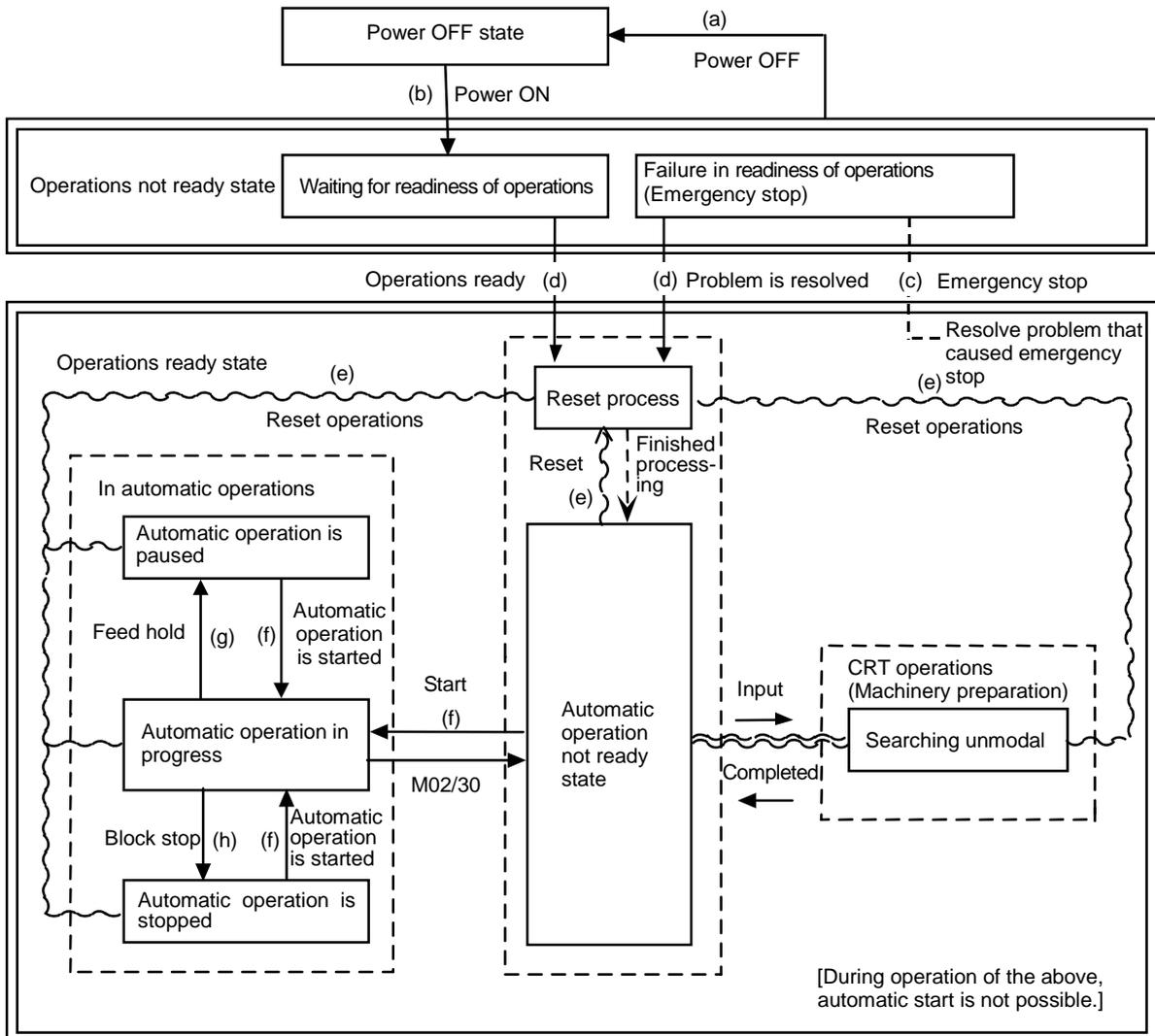


1. Operation State

1.1 Operation State Correlation Diagram

The controller operation state changes momentarily according to the program contents or signals from the operation panel or machine. The operation states are roughly classified into power OFF, not ready, and ready.

Manual mode operation is enabled in the operation complete state.



Operation state correlation diagram

1.2 Power OFF

The power OFF state means that no power is supplied to the control circuit.

(1) From any other state to power OFF (Transition of operation state correlation diagram (a))

- When POWER OFF signal is input from the machine;
- When power supplied from the machine to controller is turned OFF; or
- When power unit fuse in the controller blows.

1.3 Operations Not Ready

The operation not ready state means that the system is not ready to run because of the controller itself or the machine, even though power is supplied to the NC unit control circuit. The RUN READY lamp on the setting and display unit is OFF.

- (1) From power OFF to not ready (Transition of operation state correlation diagram (b))
 - When POWER ON signal is input from the machine.
- (2) From ready to not ready (Transition of operation state correlation diagram (c))
 - When EMG (emergency stop) is displayed on the setting and display unit; or
 - When any of the following alarms is displayed on the setting and display unit; Servo alarm, feedback alarm, excessive error alarm, watch dog, MCP alarm.

1.4 Operations Ready

The operations ready state means that power is supplied to the NC unit control circuit and the system is ready to run. The READY lamp on the setting and display unit is ON.

The state is furthermore classified into the following four:

1.4.1 Reset

The reset state means that the controller is reset.

- (1) From not ready to reset (Transition of operation state correlation diagram (d))

This state is also called initial state.

 - When about four seconds has passed after the power is turned ON.
- (2) From another ready state to reset (Transition of operation state correlation diagram (e))
 - When the RESET key on the setting and display unit is turned ON;
 - When external reset signal is input from the machine; or
 - When M02 or M30 is executed (depending on the machine specifications).

1.4.2 Automatic Operation in Progress

The automatic operation in progress state means running operations in automatic mode. The IN-AUTO OPERATION lamp on the machine operation panel is ON.

- (1) From another ready state to automatic operation in progress (Transition of operation state correlation diagram (f))
 - When the CYCLE START switch on the machine operation panel is pressed in automatic mode.

1.4.3 Automatic Operation Pause

The automatic operation pause state means that operation or motion temporarily pauses during execution of one block during the automatic operation start. The AUTO PAUSE lamp on the machine operation panel is ON and the AUTO START lamp is OFF.

- (1) From automatic operation start to automatic pause (Transition of operation state correlation diagram (g))
 - When the FEED HOLD switch on the machine operation panel is turned ON; or
 - When automatic mode input is out.

1.4.4 Automatic Operation Stop

The automatic operation stop state means that execution of one block is completed and stopped during automatic operation start. Both the AUTO START and AUTO PAUSE lamps on the machine operation panel are OFF.

- (1) From automatic operation in progress to automatic operation stop (Transition of operation state correlation diagram (h))
 - When the SINGLE BLOCK switch on the machine operation panel is turned ON and execution of the block is completed; or
 - When automatic mode input is changed to another automatic mode input.

CAUTION



Stay out of the moveable range of the machine during automatic operation. Keep hands, feet and face away from the rotating spindle.

2. Indicator Lamps

2.1 Control Unit Ready

The CONTROL UNIT READY lamp indicates that the control unit is ready to run. The lamp goes OFF at emergency stop or when an alarm occurs in the drive or operation block.

2.2 In Automatic Operation

The IN AUTO OPERATION lamp is ON from CYCLE START switch turning ON in the automatic operation mode (memory, tape, or MDI) to the program end after M02 or M30 execution, reset, or emergency stop.

2.3 Automatic Operation Start Busy

The AUTO START BUSY lamp indicates that the controller is executing control in the automatic operation mode. It is ON from the automatic operation start state entered when the CYCLE START switch is pressed in the automatic operation mode (tape, memory, or MDI) to the automatic operation start end such as the automatic operation pause busy state entered when the FEED HOLD switch is pressed or block completion stop (block stop).

2.4 Automatic Operation Pause Busy

The AUTO PAUSE lamp is ON from AUTO PAUSE switch turning on to AUTO START switch turning ON or when the mode selection switch is changed from the automatic to manual mode during the automatic operation.

2.5 Return to Reference Position

Output is executed when the controlled axis arrives at the reference position when manual reference position return, automatic reference position return, or reference position collation (check).

2.6 Alarm

The ALARM lamp goes ON when an alarm occurs during NC running.

2.7 M00

If M00 given in a program is executed during automatic operation, automatic operation stop is performed after execution of the M00 block is completed. The M00 lamp is turned ON. (This depends on PLC processing.)

2.8 M02/M30

When M02 or M30 is executed during automatic operation, the NC unit reaches the program end and the M02 or M30 lamp is turned ON. (This depends on PLC processing.)

3. Reset Switch and Emergency Stop Button

3.1 Reset Switch

The controller is reset by turning ON the RESET switch on the machine operation panel or the RESET key on the setting and display unit. When the RESET switch or key is turned ON while the controller is running, the following states are to be observed.

- (1) If a movement command is being executed, movement stops with deceleration and the remaining distance in the executing block is cleared.
- (2) If miscellaneous function such as M, S, or T is being executed, execution of the miscellaneous function is interrupted.
- (3) The active and buffer memory contents and display are cleared.
- (4) If a program error is being occurred, the program error state is cleared and the ALARM lamp goes OFF.
- (5) If the reset switch is turned ON while the input/output device is running, the power will be cut off.
- (6) When the reset switch is turned ON, the modal state will return to its original state.

3.2 Emergency Stop Button

The EMERGENCY STOP button is a red mushroom-shaped pushbutton. The unready state is set by pressing the EMERGENCY STOP button. During emergency stop, the READY lamp goes OFF and automatic operation and manual operation do not work. The controller is reset at that time.

If the EMERGENCY STOP button is pressed when a movement command is executed, the moving axis stops and all other machine motions also stop.

When the EMERGENCY STOP button is released, the READY lamp goes ON in about one second and operation enable state (READY state) is entered.

When parameter is emergency stop hold type, even if the EMERGENCY STOP button is released, the emergency stop state is held. To release the emergency stop state, turn ON the RESET switch.

When the EMERGENCY STOP LIMIT switch of each axis works, the same state as when the EMERGENCY STOP button were pressed may be entered depending on the machine specifications.



CAUTION



If the axis overruns or emits an abnormal noise, immediately press the emergency stop button and stop the axis.

4. Operation Mode

4.1 Mode Selection Switch

The MODE SELECT switch is used to determine the controller operation mode.

Jog feed mode:

Select the mode to move the controlled axis consecutively at manual feedrate.

Rapid traverse feed mode:

Select the mode to move the controlled axis consecutively at rapid traverse feedrate.

Return to reference position mode:

Select the mode to position the controlled axis at the machine reference position manually.

Incremental mode:

Select the mode to move the controlled axis for a given distance.

Handle feed mode:

Select the mode to move the controlled axis by using the manual handle.

Memory Mode:

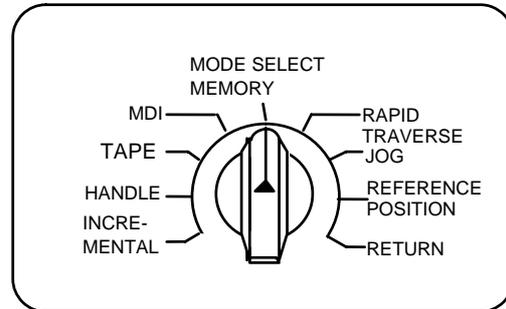
Select the mode for memory operation.

Tape mode:

Select the mode for tape operation.

MDI mode:

Select the mode for MDI operation.



(Note 1) Refer to the section 1.4 for the running state when a mode is changed to another one during automatic operation.

4.2 Jog Feed Mode

The jog feed mode enables the machine to be moved by hand consecutively at the feedrate set by using the MANUAL FEED RATE switch. The jog feed mode is started by using the FEED AXIS SELECT switch. Refer to the section 5 for the MANUAL FEED RATE switch.

Operation procedure

Using the MODE SELECT switch, select the jog mode.

Using the MANUAL FEED RATE switch, set the feedrate.
The feedrate unit is the travel distance (mm) per minute.

To move the controlled axis, turn on the FEED AXIS SELECT switch. The controlled axis is moved while the switch is turned on.
When the switch is turned off, the controlled axis stops with deceleration.

(Note 1) When the MANUAL OVERRIDE switch on the operation panel is turned ON, the override value set by using the FEED RATE OVERRIDE switch takes precedence over the feedrate set by using the MANUAL FEED RATE switch.

4.3 Rapid Traverse Feed Mode

The rapid traverse feed mode enables the machine to be moved consecutively at rapid traverse feedrate manually.

The rapid traverse feedrate can be changed its speed in four steps by using the RAPID TRAVERSE OVERRIDE switch. The rapid traverse feed mode is started by using the FEED AXIS SELECT switch.

(Note 1) Refer to the manual issued by the machine tool builder for the rapid traverse feedrate.

(Note 2) Refer to the section 5 for the RAPID TRAVERSE OVERRIDE switch.

Operation procedure

Using the MODE SELECT switch, select the rapid traverse feed mode.

Using the RAPID TRAVERSE OVERRIDE switch, set arbitrary override value.

To move the controlled axis, turn ON the FEED AXIS SELECT switch. The controlled axis is moved while the switch is turned ON.
When the switch is turned OFF, the controlled axis stops with deceleration.

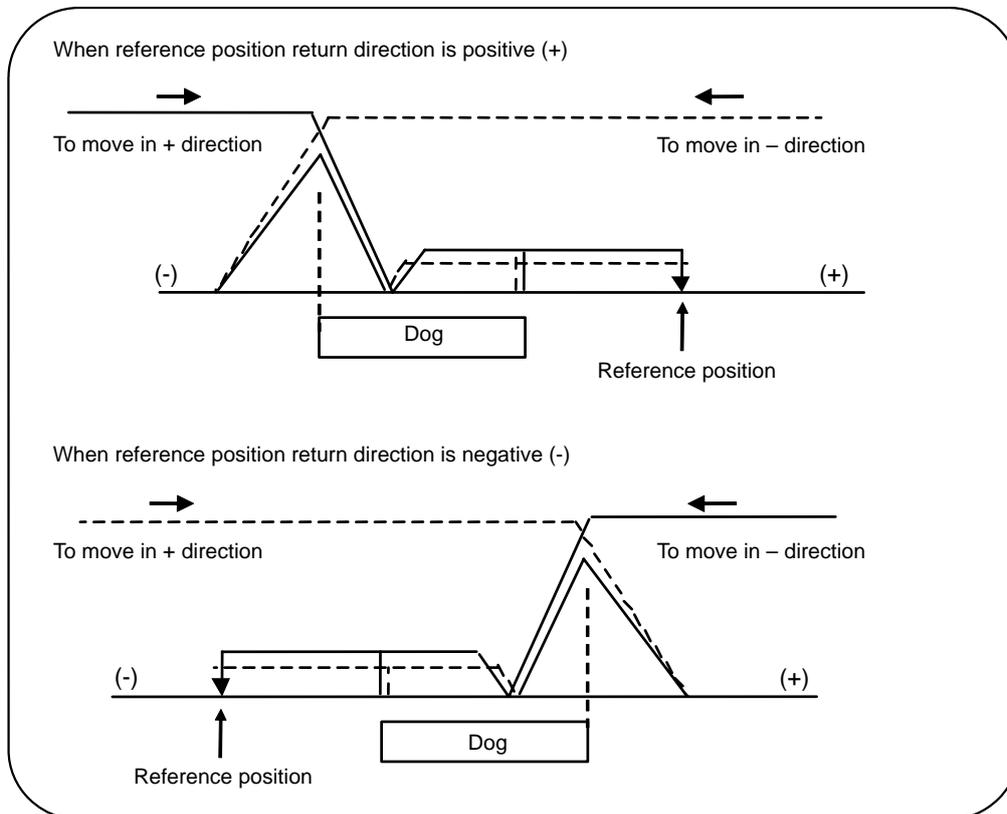
(Note 1) The override value set by using the CUTTING FEED OVERRIDE switch is not effective for the rapid traverse feedrate; when the override value is 0%, the controlled axis does not move.

4.4 Manual Reference Position Return Mode

This mode enables a controlled axis to be returned manually to the defined position unique to the machine (reference position).

The first reference position return after the NC power is turned ON becomes the dog mode. For the second or later reference position return, the dog mode or high-speed return can be selected by setting a given parameter.

Patterns of reference position return are as shown below.



Dog mode reference position return

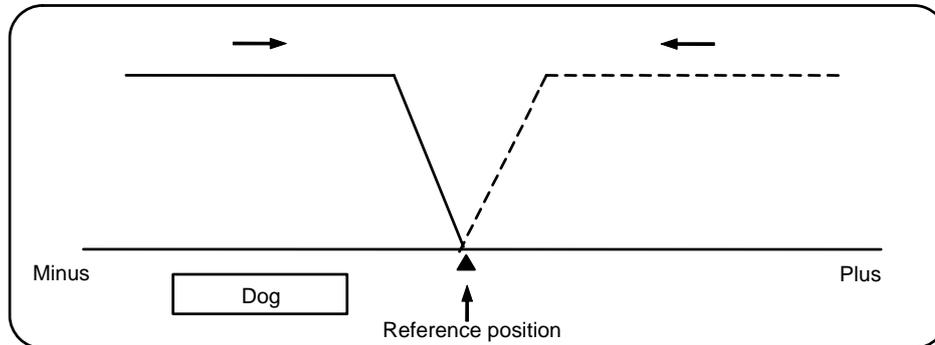
The steps below describe what happens to the controlled axis when it returns to the reference position for the first time with the power ON and with the machine in an "operations not ready state" (emergency stop is engaged or the servo alarm is ON) or when the parameters are selected in the dog mode.

- (1) The controlled axis is moved in the direction where the near point detection limit switch and dog approach each other in the reference position return mode.
- (2) When the limit switch kicks the dog, the controlled axis once stops with deceleration.
- (3) Next, the controlled axis moves to the reference position at the approach rate set in the parameter.
- (4) When it arrives at the reference position, the reference position arrival signal is output.
- (5) The reference position establishment signal is output.

High-speed reference position return

If high-speed return is set in a given parameter after dog mode reference position return is executed, then high-speed reference position return will be made.

If the return direction is erroneous in high-speed reference position return, an alarm occurs. A return is made to the reference position at the rapid traverse feedrate.

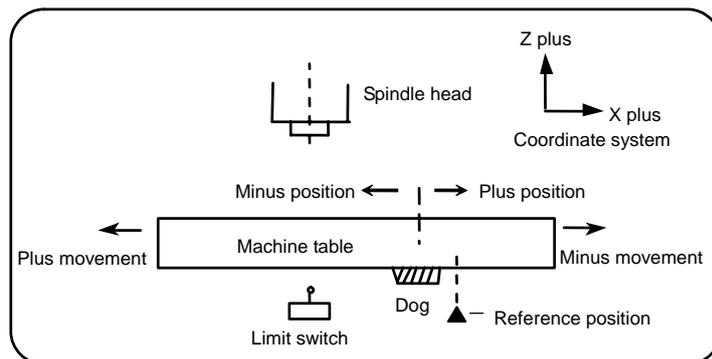


Operation procedure

Using the MODE SELECT switch, select the reference position return mode.

Using the RAPID TRAVERSE OVERRIDE switch, set arbitrary override value.

Check the current machine position.



The machine position depends on whether the near point detection limit switch is on the plus or minus side with the near point dog on the machine table as illustrated above as the reference. When the limit switch exists on the dog, move to either plus or minus.

Using the FEED AXIS SELECT switch, move the machine.
If the limit switch exists in the minus direction as illustrated above, turn ON a plus FEED AXIS SELECT switch.

For dog mode reference position return, turn ON the FEED AXIS SELECT switch (+ or -) in the direction where the dog and limit switch approach each other.
For high-speed reference position return, turn ON the FEED AXIS SELECT switch (+ or -) in the direction that the spindle head approaches the reference position.
Hold the FEED AXIS SELECT switch ON during reference position return until the machine passes by the dog (dog mode) or the REFERENCE POSITION ARRIVAL lamp goes ON (high speed return).

4.5 Incremental Feed Mode

The incremental feed mode enables the controlled axis to be moved at a given distance selected by using the HANDLE/INCREMENTAL MAGNIFICATION switch at the manual feedrate when the FEED AXIS SELECT switch is ON.

Operation procedure

Using the MODE SELECT switch, select the incremental feed mode.

Using the HANDLE/INCREMENTAL MAGNIFICATION switch set a travel distance.

The controlled axis selected by turning ON the FEED AXIS SELECT switch once is moved at a given distance.

4.6 Handle Feed Mode

The controlled axis can be moved by turning the manual handle.

The travel distance per graduation of the handle depends on how the HANDLE/INCREMENTAL MAGNIFICATION switch is set.

The axis that can be moved by using the manual handle is determined by setting the HANDLE FEED AXIS SELECT switch.

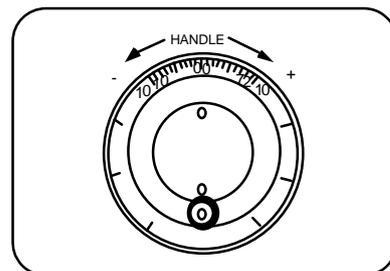
Operation procedure

Using the MODE SELECT switch, select the handle feed mode.

Using the HANDLE FEED AXIS SELECT switch, select the controlled axis to be moved.

Using the HANDLE/INCREMENTAL MAGNIFICATION switch, set the travel distance per graduation of the handle.

Move the axis by turning the HANDLE in arbitrary direction.



4.7 Memory Mode

In the memory mode, a work program registered in memory is called and automatic operation is executed.

Operation procedure

Call the work program for memory operation by using the setting and display unit.

Check whether or not the work program is called normally.

Using the MODE SELECT switch, select the memory mode.

Set an arbitrary override value by using the switch RAPID TRAVERSE OVERRIDE, FEED RATE OVERRIDE, SPINDLE OVERRIDE. Normally, set the value to 100%.

Automatic operation is started by turning ON the CYCLE START switch.

The CYCLE START switch becomes effective when it is once turned ON, then OFF.

To temporarily stop machine motion, turn ON the FEED HOLD switch. The controlled axes being moved stop with deceleration.

When machine motion is stopped by using the FEED HOLD switch, automatic operation will be restarted by turning ON the CYCLE START switch.

Memory operation terminates when M02 or M30 in the program is executed. The M02 or M30 lamp on the machine operation panel is turned ON.

To repeat executing the same program, input the rewind signal by reset & rewind with PLC program.

To forcibly terminate automatic operation, turn ON the RESET switch.

CAUTION

-  Carry out dry run operation before actually machining, and confirm the machining program, tool offset amount and workpiece coordinate system offset amount.

4.8 MDI Operation Mode

In the MDI operation mode, automatic operation is executed by using a program set on the setting and display unit MDI program editing screen.

Operation procedure

The MDI operation follows the memory operation.

Set data on the setting and display unit MDI program editing screen.

Using the MODE SELECT switch, select the MDI mode.

The following steps are the same as the memory operation steps. Refer to 4.7.

(Note 1) When using a 2-part system, the MDI data operation can be switched with the parameters.

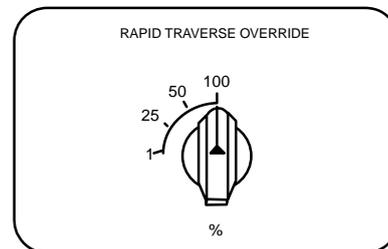
#1050 MemPrg	Details
0, 2, 4, 6	The common MDI program for the part system is run in each part system.
1, 3, 5, 7	MDI programs for each part system are run in each part system.

5. Operation Panel Switches in Operation Mode

5.1 Rapid Traverse Override

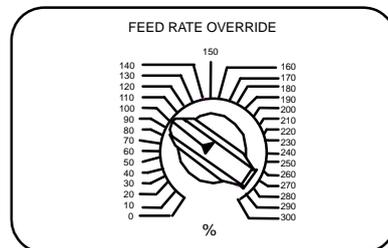
Use the RAPID TRAVERSE OVERRIDE switch to override the rapid traverse feedrate in automatic or manual operation.
RAPID TRAVERSE OVERRIDE is applicable to the following:

Automatic operation: G00, G27, G28, G29, G30
Manual operation: Rapid traverse,
reference position return,
incremental feed



5.2 Cutting Feed Override

Use the FEED RATE OVERRIDE switch to override the feedrate in automatic operation (G01, G02, or G03 F command) or the manual feedrate of jog feed in manual operation in 10% units in the range of 0% to 300%. FEED RATE OVERRIDE is also applicable to the dry run rate in automatic operation.

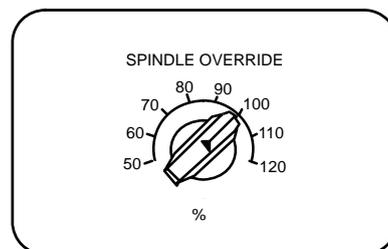


(Note 1) The dry run rate is the movement rate set by using the MANUAL FEED RATE switch by overriding the programmed feedrate in automatic operation.

(Note 2) Refer to the section 6.5 for feedrate override applied to manual feedrate.

5.3 Spindle Override

This function applies override to the rotation speed of a spindle or mill spindle assigned by the machining program command during automatic operation or by manual operation. There are two types of override.



(1) Type 1 (code method)

Using an external signal, override can be applied to the commanded rotation speed of a spindle or mill spindle in 10% increments from 50% to 120%.

(2) Type 2 (value setting method)

Using an external signal, override can be applied to the commanded rotation speed of a spindle or mill spindle in 1% increments from 0% to 200%.

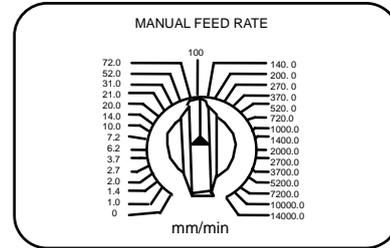
(Note 1) Selection between type 1 and type 2 can be designated by PLC program processing.

5.4 Manual Feedrate

Use the MANUAL FEED RATE switch to set the feedrate in jog feed mode during manual operation. The feedrate can be selected among 31 steps from 0 to 14000.0 mm/min.

When the MANUAL OVERRIDE switch (interrupt switch) is turned ON, the override value set by using the FEED RATE OVERRIDE switch takes precedence over the value set by using the MANUAL FEED RATE switch.

Manual feedrate (mm/min)				
0.	7.2	72	720	7200
1.0	10.0	100	1000	10000
1.4	14.0	140	1400	14000
2.0	20.0	200	2000	
2.7	27.0	270	2700	
3.7	37.0	370	3700	
5.2	52.0	520	5200	



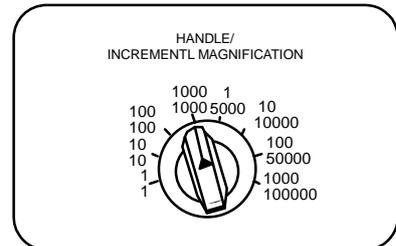
5.5 Handle/Incremental Feed Magnification

Use the HANDLE/INCREMENTAL MAGNIFICATION switch to set the travel distance commanded when manual handle feed or incremental feed is made.

The travel distances for each axis are listed below.

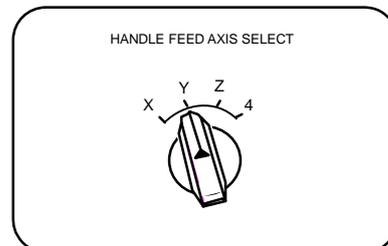
When B or C is set to the parameter "#1003 iunit", up to 1000 only can be validated.

Movement amount ("#1003 iunit"=B or C)	Movement amount (Other than "#1003 iunit"=B or C)
1	1
10	10
100	100
1000	1000
5000	5000
10000	10000
50000	50000
100000	100000



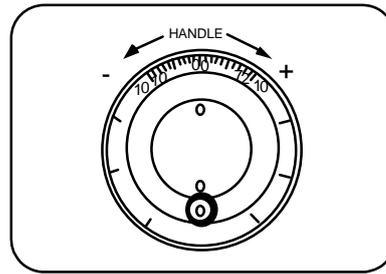
5.6 Handle Feed Axis Selection

Use the HANDLE FEED AXIS SELECT switch to select the axis moved by handle operation when the handle mode is selected.



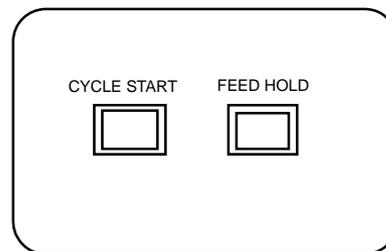
5.7 Manual Pulse Generator

In the manual handle mode, fine feed of the machine can be made by turning the manual pulse generator. The manual pulse generator has 100 graduations per revolution and outputs one pulse per graduation. The travel distance per pulse is set by using the HANDLE/ INCREMENTAL MAGNIFICATION switch.



5.8 Cycle Start and Feed Hold

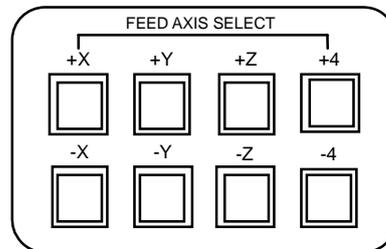
Use the CYCLE START switch to start automatic operation (memory, tape, or MDI). Automatic operation is executed by turning ON the switch. Use also the switch for restart from stop by the FEED HOLD switch or the automatic operation stop state.



The CYCLE START switch becomes effective when the switch is turned ON, then OFF. Use the FEED HOLD switch to temporarily stop automatic operation (for example, deceleration stop of the control axis during automatic operation). To restart operation, use the CYCLE START switch.

5.9 Feed Axis Selection

Use the FEED AXIS SELECT switch to start the controlled axis during manual operation. While the FEED AXIS SELECT switch is held ON, the selected controlled axis is moved. When the switch is turned OFF, the controlled axis movement stops.



6. Operation Panel Switch Functions

6.1 Chamfering

Chamfering in the thread cutting cycle can be validated/invalidated using an external switch.

6.2 Miscellaneous Function Lock

- (1) M, S, T or B function execution can be ignored by turning ON the MISCELLANEOUS FUNCTION LOCK switch.
- (2) M, S, T or B function BCD output is made, but the start signal is not output.
- (3) If the MISCELLANEOUS FUNCTION LOCK switch is changed during command execution, automatic operation stops after the block being executed is terminated. Then, it becomes effective.

6.3 Single Block

- (1) When the SINGLE BLOCK switch is turned ON, automatic operation stops after the block being executed is terminated. That is, automatic operation stops after one program block is executed.
- (2) The single block stop point in the fixed cycle mode is fixed according to the fixed cycle.

6.4 Z Axis Cancel

- (1) By turning ON the "Z AXIS CANCEL" switch, NC commands can be executed for the movement by automatic operation, without moving the machine in the Z axis only.
The Z axis does not move, but the current position display of the setting display unit is incremented.
- (2) The Z axis moves by manual operation even when the "Z AXIS CANCEL" switch is turned ON.
- (3) If the "Z AXIS CANCEL" switch is changed over during automatic operation, the new switch setting is validated when the automatic operation stops after the block being executed finishes.
- (4) In reference position returns (G28, G30), the axis is controlled to the middle point in a canceled Z axis state, but this state is ignored after the middle point.

6.5 Dry Run

- (1) When the DRY RUN switch is turned ON, the feedrate set by using the MANUAL FEED RATE switch takes precedence over the programmed feedrate (F).

6.6 Manual Override

- (1) When the MANUAL OVERRIDE switch is turned ON, the override value set by using the FEED OVERRIDE switch takes precedence over the value set by using the MANUAL FEED RATE switch.
- (2) The override value set by using the FEED OVERRIDE switch also takes precedence over the dry run during automatic operation.
- (3) Manual override becomes effective immediately when the switch is turned ON.

6.7 Override Cancel

- (1) When the OVERRIDE CANCEL switch is turned ON, the programmed F command value takes precedence over the override value set by using the FEED RATE OVERRIDE switch.
- (2) It is not effective for manual override.

6.8 Optional Stop

- (1) If M01 is programmed, the machine automatically stops by turning ON the OPTIONAL STOP switch. When the switch is OFF, M01 is ignored and the machine does not stop.
- (2) The machine stops after the M01 block is executed.

6.9 Optional Block Skip

When the OPTIONAL BLOCK SKIP switch is turned ON, a block which begins "/" (slash) or "/n" code is skipped; when the switch is OFF, the block is executed. This enables the operator to specify whether or not a block beginning with "/" (slash) or "/n" code is executed.

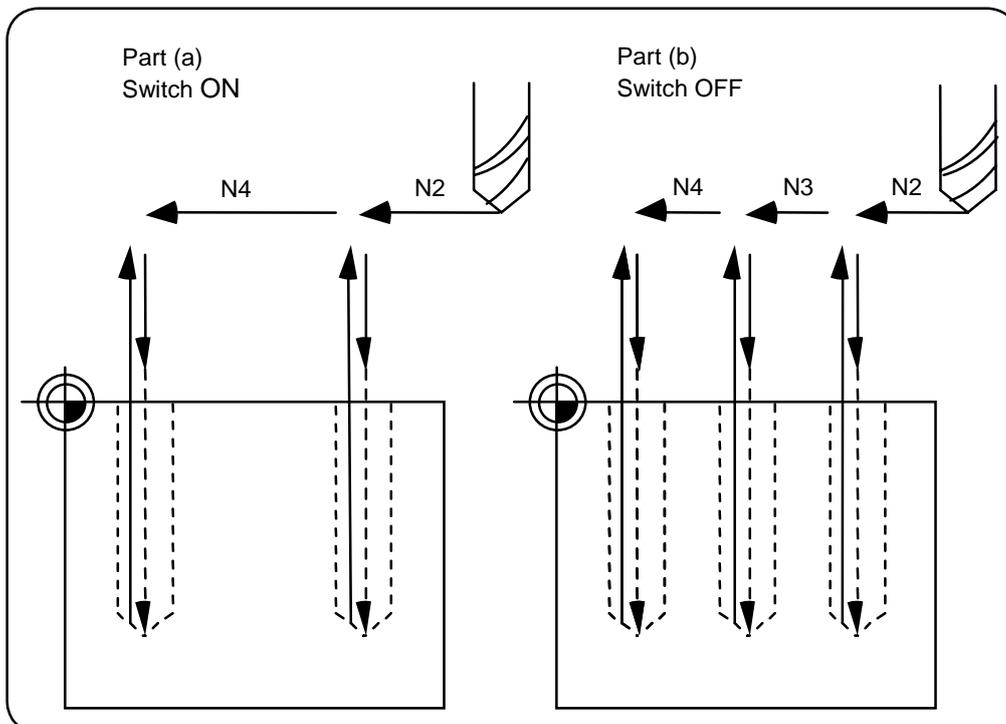
A block which has a "/" or "/n" code in not the head but the middle is operated according to the parameter "#1226 aux10/bit1 (optional block skip type) setting.

0: Enable only at the beginning of a block.

1: Enable in the middle of a block, as well as at the beginning of the block.

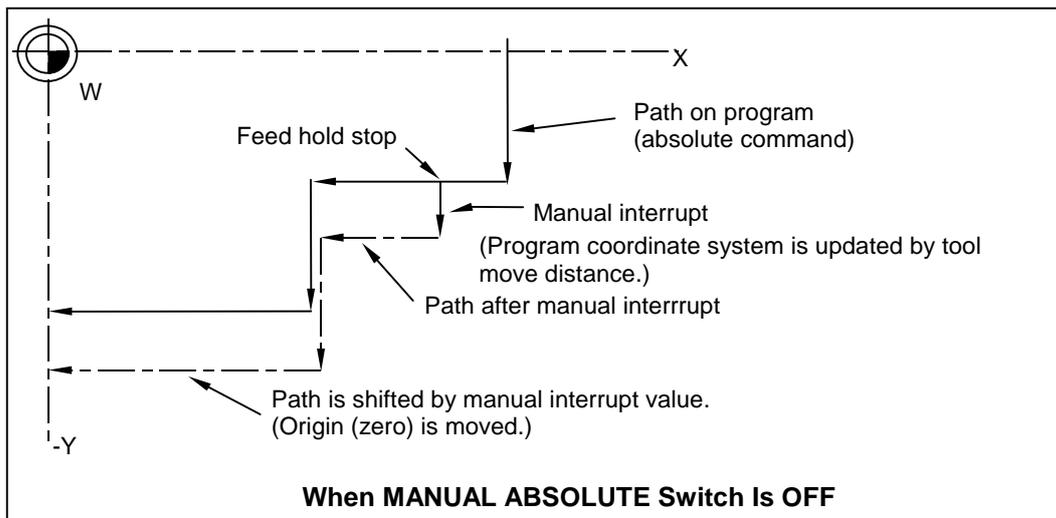
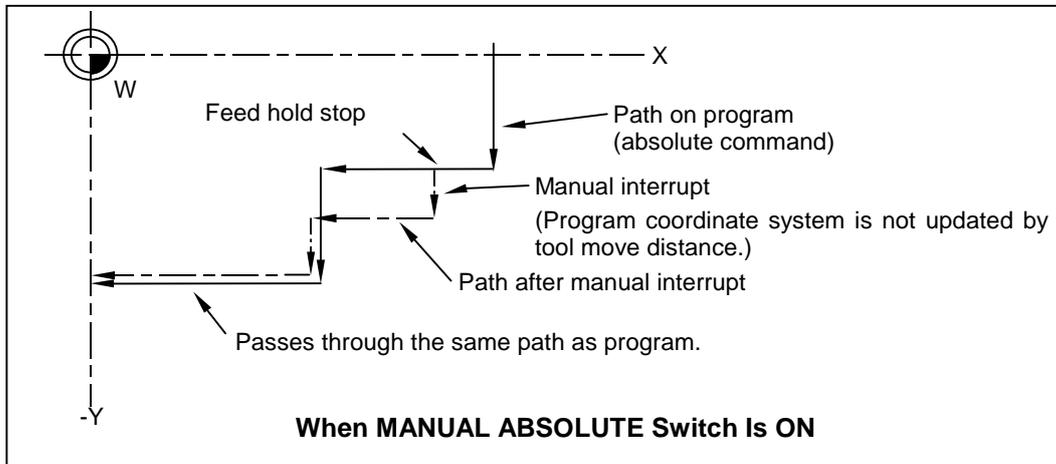
(Example) When machining two parts as illustrated below, if the following program is prepared and the machining is done by turning ON the OPTIONAL BLOCK SKIP switch, part (a) is provided; if the machining is done by turning OFF the switch, part (b) is provided:

```
Program  N1 G54 ;
         N2 G90 G81 X50. Z-20. R3. F100 ;
         /5 N3 X30. ;
         N4 X10. ;
         N5 G80 ;
         M02 ;
```



6.10 Manual Absolute

When the MANUAL ABSOLUTE switch is turned ON, while the program coordinate system is not updated, the coordinate position is updated by the distance in which the tool has been moved manually. If the switch is OFF, the program coordinate system is updated when the tool is manually moved.



6.11 Mirror Image

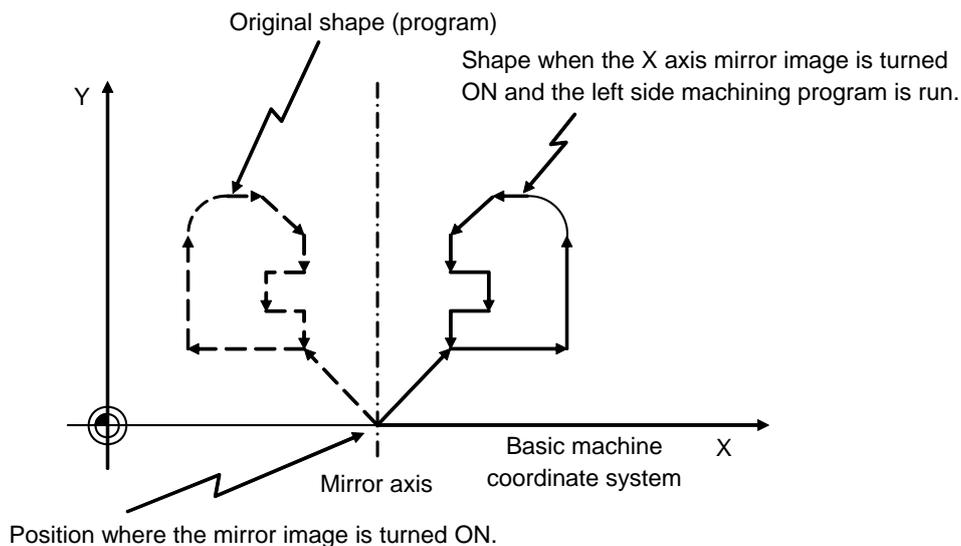
6.11.1 Outline

When cutting a shape having bilateral symmetry, this function can be used to machine either the right or left side of the shape from the program for the other side, thus saving programming time. When cutting a shape having upper and lower symmetry, this function can be used to machine either the upper or lower side of the shape from the program for the other side.

The mirror image function is effective in this case.

For example, when there is a program for machining the left side of the shape shown in the following drawing, a symmetrical shape to the left side shape can be machined on the right side by applying the mirror image function and running the same program.

Be sure the machining program starts and finishes at the mirror center.



Mirror image includes external input mirror image, parameter setting mirror image, programmable mirror image, etc. The external input mirror image and parameter setting mirror image are explained here. (Refer to the Programming Manual for information on programmable mirror image.)

- External input mirror image : The mirror image function is controlled with the MIRROR IMAGE switch installed on the machine operation panel.
- Parameter setting mirror image : The mirror image function is controlled by the parameter setting on the NC unit screen.

(Note) Refer to the Operation Manual issued by the machine tool builder for information on the operation panel MIRROR IMAGE switch.

The mirror image can be set to ON/OFF for each axis in both external input mirror image and parameter setting mirror image.

6.11.2 Detailed Description

The external input mirror image

- (1) Before carrying out mirror image machining, move the axis to the mirror center using a mode such as MDI or JOG.
- (2) After positioning the axis on the mirror center, turn ON the MIRROR IMAGE switch.
- (3) Movement for the movement commands subsequently issued in the respective program run and MDI modes is carried out to positions symmetrical with the mirror center, and the shape is machined.
- (4) Return to the mirror image center.
- (5) Turn OFF the MIRROR IMAGE switch to cancel the mirror image function.

The parameter setting mirror image

The parameter setting mirror image operation is equivalent to that of the external input mirror image, except that the "1" or "0" is set in the parameter "#8211 Mirror image" instead of using a MIRROR IMAGE switch.

0: Mirror image OFF

1: Mirror image ON

This is used when there is no external switch for mirror image operation on the machine operation panel, etc.

Mirror image operation switching

The type of mirror image operation can be select by the parameter "#1271 ext07/bit0 (Mirror image operation).

0: Type 1

- The program mirror image, external mirror image, and parameter mirror image are exclusive to each other.
- An increment command moves the image to the position indicated by the move amount with the sign inverted.

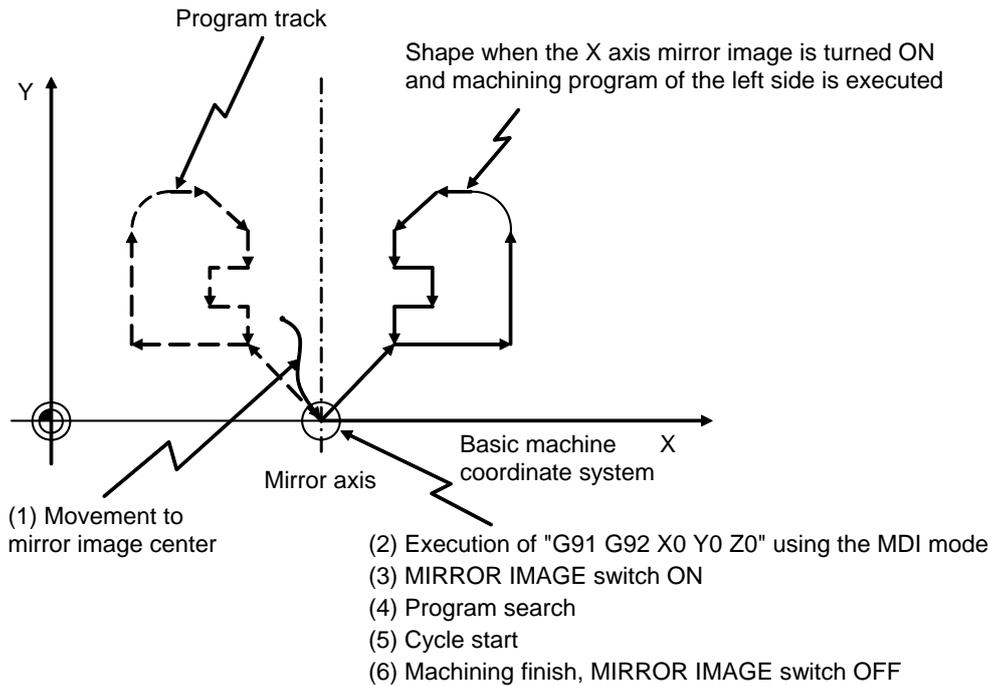
1: Type 2

- Mirror image operation is enabled when the program mirror image (G51.1) command is issued or when the external signal or parameter is ON.
- An increment command moves the image to the position determined by applying the mirror image to the absolute program coordinates.

Example

<Example of operation with an external switch>

- (1) Move the axis to the mirror image center using a the JOG mode.
- (2) Execute "G91 G92 X0 Y0 Z0" using the MDI mode.
- (3) Turn ON the MIRROR IMAGE switch.
- (4) Search for the program to which mirror image will be applied.
- (5) Press the CYCLE START switch to start the program.
- (6) Turn OFF the MIRROR IMAGE switch when the machining is finished.



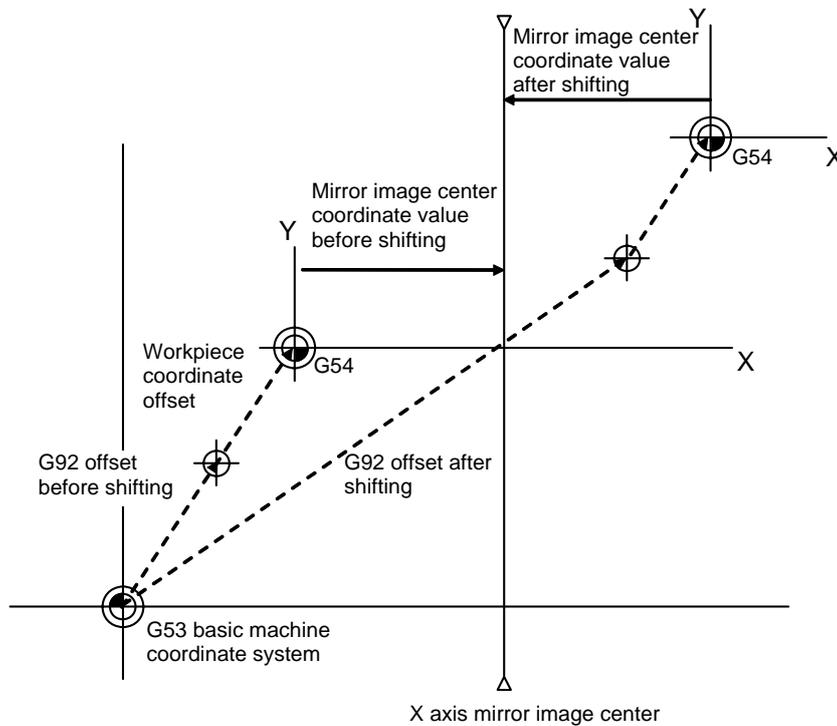
(Note) If an "NC reset" or "Machining program restart" is carried out during operation with MIRROR IMAGE switch ON, the mirror image center will move to the "Restart position" and the coordinate system will shift.

6.11.3 Combination with Other Functions

(1) Coordinate system setting, workpiece coordinate system

As for the mirror image center, the position in which an external signal or the parameter was turned ON is the center coordinates value.

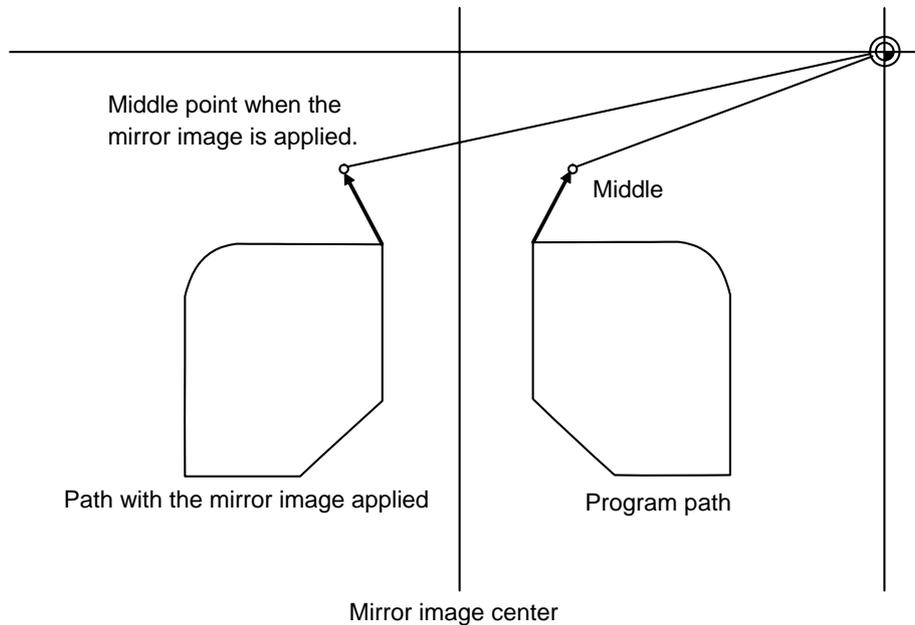
External input mirror image and parameter setting mirror image are processed on the basic machine coordinates. Therefore, the mirror image center is not shifted by the coordinates system setting (G92) and the workpiece coordinates change.



For the mirror image of external input mirror image and parameter setting mirror image, the coordinate value on the basic machine coordinate system is not shifted. However, the coordinate value on the local coordinate system is shifted according to the coordinate system specified with G92.

(2) Reference position return command (G28, G30) during mirror image

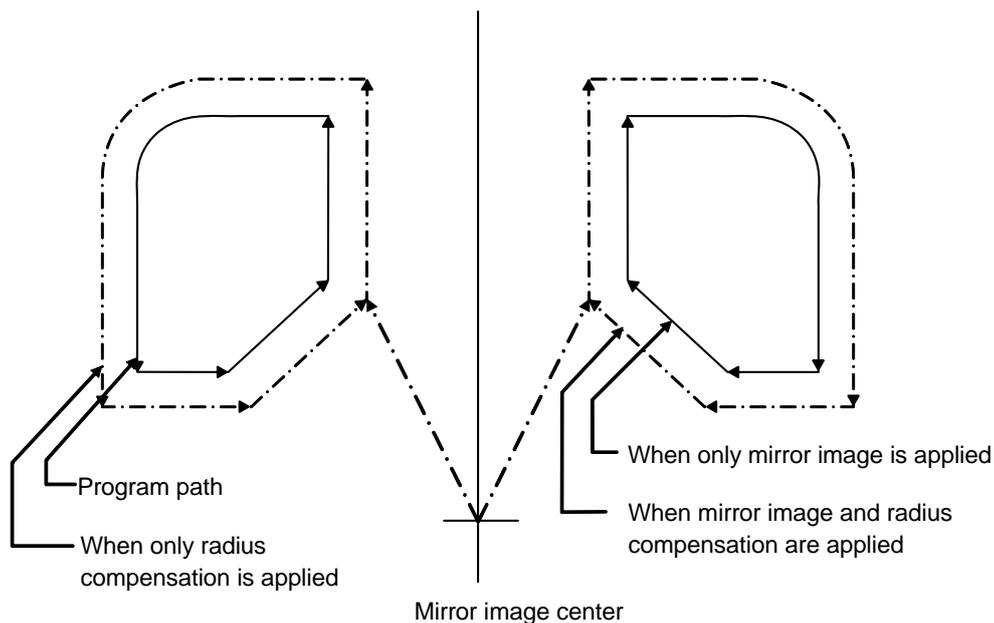
When a reference position return command (G28, G30) is issued during mirror image, the mirror image is valid in the operation until the middle point, but is invalid in the operation from the middle point to the reference position.

**(3) Reference position return command (G29) from the zero point**

When a reference position return command (G29) is issued from the reference position, the middle point becomes the middle point passed through during the previous reference position return command (G28, G30), unrelated to the current mirror state. The axis moves following the current mirror state (ON/OFF) for operations after the middle point.

(4) Radius compensation commands (G41, G42) during mirror image (M system)

Because the mirror image is processed after the radius compensation (G41, G42) is applied, the cutting is carried out as follows.



(5) Scaling

The mirror image is applied for the scaled shape.

(6) Machine coordinate system selection (G53)

The mirror image is not applied for positioning by a G53 command.
Issue the next movement command in the G90 mode.

(7) Fixed cycle (G76, G87) (M system)

The mirror image is not applied to shift operations in the XY direction for G76 and G87 commands.
The mirror image is applied to positioning operations.

(8) Handle interrupt

When the manual ABS is ON, the handle interrupt amount is reflected on the workpiece coordinate system counter, and the workpiece coordinate system does not shift. Because of that, the mirror center also does not shift.

When the manual ABS is OFF, the handle interrupt amount is not reflected on the workpiece coordinate system counter. The amount is held as an interrupt amount, and the mirror center shifts.

(9) Arc, tool compensation

If mirror image is applied on only one axis in the designated plane, the rotation direction and compensation direction will be reversed for the arc or tool compensation and coordinate rotation, etc.

6.11.4 Precautions

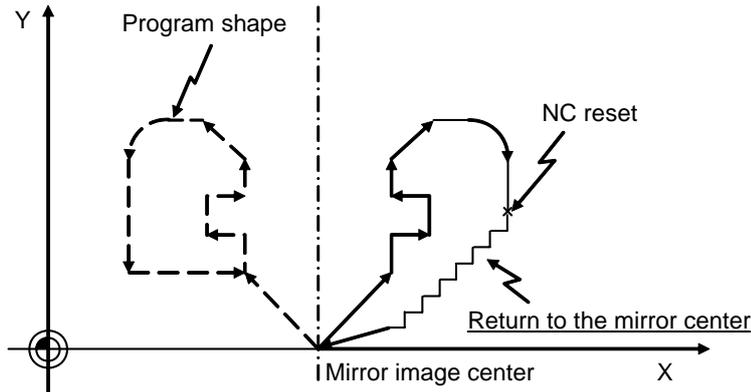
(1) Reset and mirror image cancel

The mirror image center is canceled by the NC reset (including M02, M30 internal resets). The next time the mirror image center will be established is when an automatic start is carried out.

This operation enables the mirror image center to be changed with the mirror image ON using a simple operation, so that another shape can be machined after the machining finishes.

Be sure to move the axis to the mirror image center with the manual handle, etc. before carrying out an automatic start when restarting the same machining program.

Be sure the final command M02 (or M30) position is at the mirror image center (program start position) when repeating the same machining program.



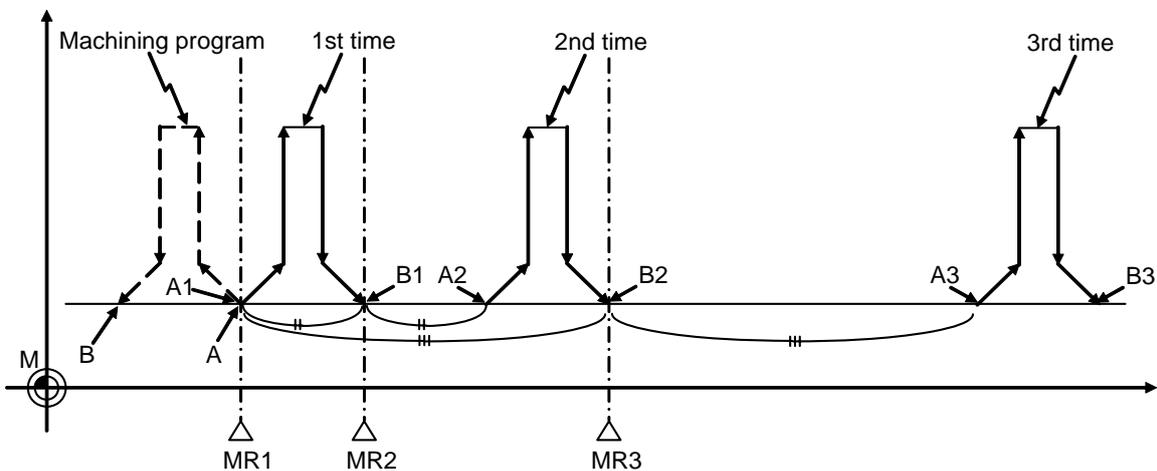
< Program example when repeating the program >

```

G90 G0 X0 Y0 ; ..... Program start position
      X___ Y___ ;
      :
G0 X0 Y0 ; ..... Return to program start position.
M02 ; ..... NC reset
    
```

If the NC is reset at a point other than the mirror image center, and the same machining program is restarted, the mirror image center will move to that location. The coordinate system will shift.

If the machining program is repeated without returning to the mirror image center (program start position), the positioning point shifts from A1 → A2 → A3 as shown in the following drawing, and does not go to the desired position. This means the end point shifts from B1 → B2 → B3. At this point the M02 reset and automatic start is carried out, because the mirror image center moves from MR1 → MR2 → MR3.



< Example of a machining program in which the mirror image center shifts >

```
G90 G0 X0 Y0 ; ..... A positioning point
    G1 X___ Y___ ;
        :
        X-100. Y0 ; ..... B end point
    M02 ; ..... NC reset
```

(2) Mirror image operation at reset

The operation type of the external input mirror image and parameter setting mirror image can be select by the parameter "#8124 Mirr img at reset". If the mirror image center is not changed when resetting, set "1".

- 0: The current mirror image is canceled, and new mirror image will start with the machine position at reset as the mirror center.
- 1: The mirror center is kept to continue the mirror image. If it is necessary to change the mirror image center, command the mirror image again with new mirror image center after the mirror image is canceled once.

(3) Incremental commands after the mirror image is canceled

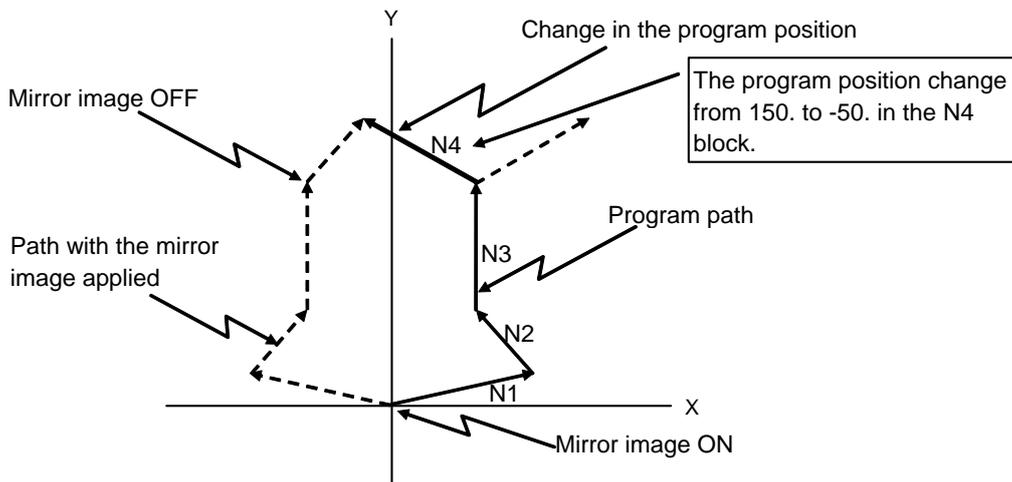
When the mirror image is canceled at a point other than the mirror image center, the program position and machine position will be in a shifted state as shown in the following drawing. To return this state to normal, carry out a process which returns the program position to the mirror image OFF position.

Because of this, the program position in the cycle start may change greatly. Cancel the mirror image at the mirror image center, or issue an absolute value positioning or reference position return command (G28, G30) once after canceling the mirror image.

The mirror state in the NC can be confirmed with the "MR" display.

Even if the NC is reset, "MR" is displayed for mirror image valid axis immediately before reset. This is so the MIRROR IMAGE switch ON/OFF can be monitored.

The "MR" display disappears when the MIRROR IMAGE switch is turned OFF and the CYCLE START is pressed.



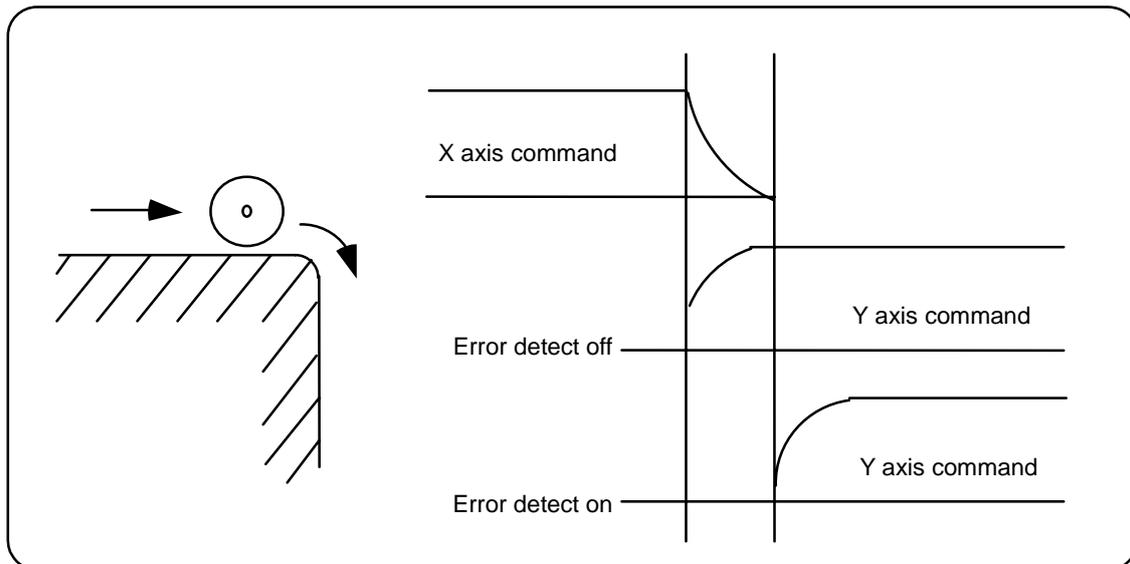
			Machine position		Program position	
			X axis	Y axis	X axis	Y axis
X axis mirror image ON	N1	G91 G00 X250. Y50. ;	-250. MR	50.	250.	50.
	N2	G01 X-100. Y100. ;	-150. MR	150.	150.	150.
	N3	Y200. ;	-150. MR	350.	150.	350.
X axis mirror image OFF	N4	X100. Y100. ; M02 ;	-50	450.	-50.	450.

6.12 Error Detect

For positioning (G00), machine deceleration check is made before next block movement is started. For cutting (G01, G02, or G03), the next block is started before the machine reaches the movement command end point. Thus, the corner part is slightly rounded.

To prevent rounded corners, turn ON the error detect signal. This will cause the machine to decelerate until the remaining distance falls below the value of the parameter. The next block command is stopped during this time. This function is equivalent to G09 in the program.

The parameter that is used by the error detect switch and the G09 command for determining the remaining distance after deceleration for moving to the next command can be set with the setting and display unit.



6.13 Follow-up Function

The follow-up function monitors machine motion in the emergency stop state and reflects it in the current position and workpiece coordinates. Thus, the work program can be continued without re-executing a reference position return after emergency stop.

6.14 Axis Removal

When the machine receives the axis removal signal, that axis no longer becomes the controlled axis. Accordingly, the alarm related with axis, such as the stroke end axis, and the servo alarms (excessive errors, lack of signal, drive alarm, etc.) will be ignored. At the same time, the axis will become interlocked.

(Note) This cannot be used for the absolute position detector specification axis.

6.15 F 1-digit Feed

- (1) When the programmed feed rate has been issued as an F 1-digit command, the feed rate can be increased or reduced by turning the manual handle when turning ON the "F 1-DIGIT FEED" switch. The feed rate cannot be changed by the 2nd and 3rd handles.
- (2) The increase/decrease amount per handle gradient is expressed with the following equation.

$$\Delta F = \Delta P \times \frac{FM}{K}$$

ΔP : Handle pulse (\pm)
 FM : Upper limit of F 1-digit feedrate (#1506 F1_FM)
 K : F 1-digit feedrate change constant (#1507 F1_K)

(Example) For a 10mm/min. increase/decrease amount per handle gradient:

If F max = 3600mm/min :

$$\Delta F = 10 = 1 \times \frac{3600}{K} \quad \text{Thus, } K = 360.$$

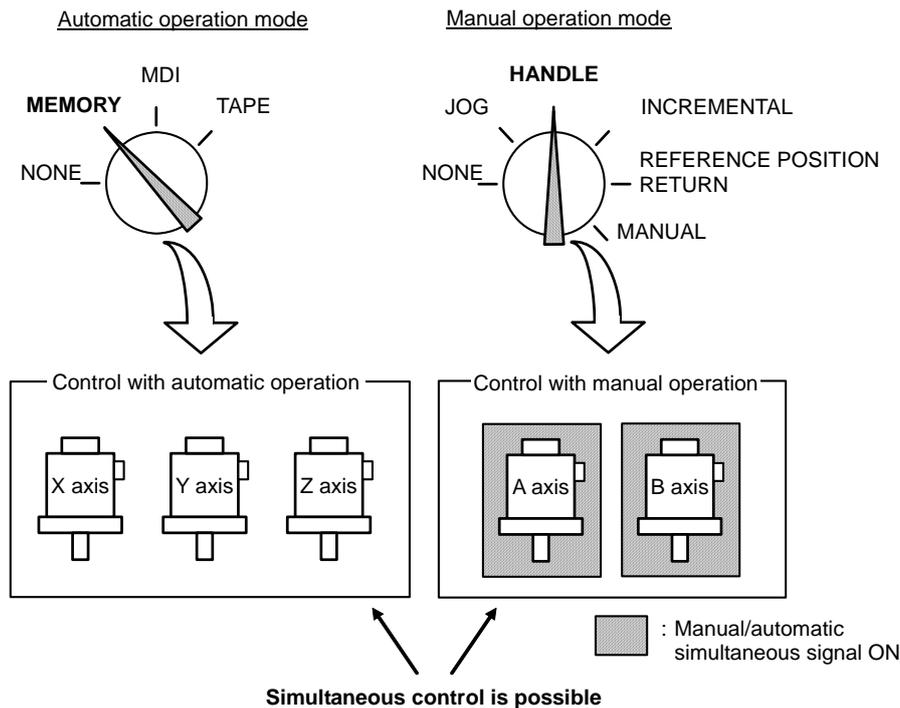
- (3) The conditions for validation are as follows:
- (a) The operation must be automatic (program run, MDI).
 - (b) The operation must be in automatic start.
 - (c) The operation must be in cutting feed, and F 1-digit feedrate command must be issued.
 - (d) The F 1-digit feed parameter must be ON.
 - (e) The F 1-digit feed switch must be ON.
 - (f) The operation must not be in machine lock.
 - (g) The operation must not be in dry run.

6.16 Manual/Automatic Synchronous Feed

6.16.1 Outline

Using this function, a random axis can be moved with manual operation even while another axis is moving with automatic operation. One automatic operation mode and one manual operation mode are selected with the PLC program and the axis to be moved with manual operation is selected. (This axis is hereafter called the "manual/automatic simultaneous valid axis".)

If the axis being moved with automatic operation is selected as the manual/automatic simultaneous valid axis, or if a movement command is issued with automatic operation to an axis already selected as the manual/automatic simultaneous valid axis, interlock will be applied on the movement by automatic operation, and the movement with manual operation will have the priority.



6.16.2 Conditions for Validating Manual/Automatic Synchronous Feed

This function is valid when the following conditions are established.

- This function's option is added.
- One automatic operation mode and one manual operation mode are selected.
(The manual operation mode can be selected during automatic operation. Normally, if the operation mode is selected in duplicate, the alarm "T01 Operation mode duplicated" will occur. However, this will not occur if this function's option is added.)
- The manual/automatic simultaneous valid axis is selected.
(The "Manual/Automatic simultaneous valid n-th axis" signal is ON for the axis to be controlled with manual operation.)

The manual/automatic simultaneous valid axis (axis for which "Manual/Automatic simultaneous valid n-th axis" signal is ON) can be controlled with the selected manual operation mode.

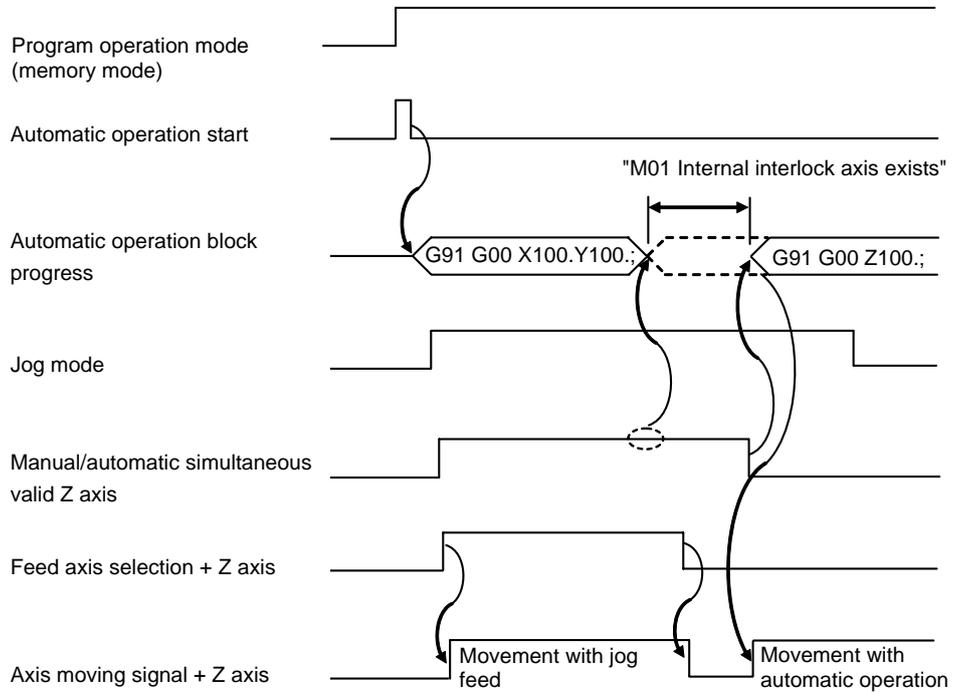
6.16.3 Operation during manual/automatic simultaneous operation

- (1) The manual operation and automatic operation feedrates are independent. The acceleration/deceleration mode (rapid traverse/cutting feed) and time constants are also independent.
- (2) The rapid traverse override is valid for both automatic operation and manual operation. The cutting feed override and 2nd cutting feed override are valid only for automatic operation. However, these will also be valid during manual operation if the manual override method selection signal is ON. Override cancel is valid only for automatic operation.
- (3) Manual interlock is valid for the manual/automatic simultaneous valid axis, and automatic interlock is valid for the axis in automatic operation.
- (4) Manual machine lock is valid for the manual/automatic simultaneous valid axis, and automatic machine lock is valid for the axis in automatic operation.
- (5) The signals in cutting feed and in rapid traverse will follow the automatic operation movement mode.
- (6) Movement of the manual/automatic simultaneous valid axis will not stop with the single block stop or feed hold stop.
- (7) If the manual/automatic simultaneous valid axis reaches the stored stroke limit or stroke limit, the axis moving in automatic operation will also stop immediately, and the feed hold state will be entered. In this case, automatic operation can be restarted by canceling the stored stroke limit or stroke limit with manual operation.
- (8) Even if the automatic operation handle interrupt option is added and the handle mode is selected for the manual operation mode, the manual/automatic simultaneous function will be the priority for the axis for which the "Manual/Automatic simultaneous valid n-th axis" signal is ON.
- (9) The speed display indicates the following speeds according to the parameter "#1125 real_f".

#1125 real_f Function	1	0
Manual/automatic simultaneous operation	Composite manual and automatic speeds (The actual feedrates are independent.)	Automatic F command speed priority (Manual command speed when there is no axis moving with automatic operation)
Only manual operation	Manual actual feedrate	Manual command speed
Only automatic operation	Automatic actual feedrate	Automatic F command speed

6.16.4 Operation When Automatic Operation Commands and Manual/Automatic Simultaneous Valid Selection are Issued for Same Axis

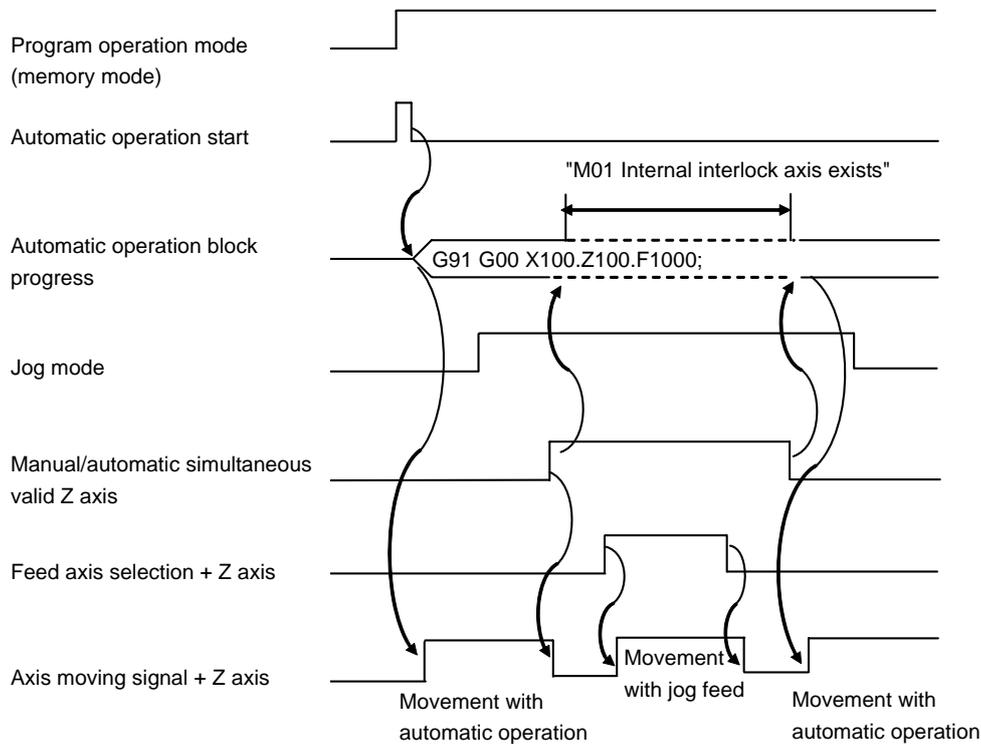
- (1) If a movement command with automatic operation is issued to a manual/automatic simultaneous valid axis, the alarm "Internal interlock axis exists" will occur. Automatic operation will stop in the interlock state until the "Manual/Automatic simultaneous valid n-th axis" signal turns OFF. The manual/automatic simultaneous valid axis can be controlled in the manual operation mode during this time.



- (2) If the "Manual/Automatic simultaneous valid n-th axis" signal is turned ON for an axis moving with automatic operation, the alarm "Internal interlock axis exists" will occur, and the operation will decelerate to a stop immediately. Automatic operation will stop in the interlock state until the "Manual/Automatic simultaneous valid n-th axis" signal turns OFF. The manual/automatic simultaneous valid axis can be controlled in the manual operation mode during this time.

Note that the interlock will be applied even during a tap modal.

Interlock will also be applied during synchronous tap cutting, and the spindle will also be synchronized to the manual drilling axis.



- (3) When the manual/automatic simultaneous valid axis is moving by manual, the G92 command from the automatic operation will be executed after the axes, including this axis, have stopped.
- (4) When several axes are interpolated and moving with automatic operation, all axes will stop if the "Manual/Automatic simultaneous valid n-th axis" signal turns ON for even one axis. When moving the axes independently such as during automatic reference position return, or during rapid traverse when the parameter "#1086 G00 non-interpolation" is ON, only the axis for which the "manual/automatic simultaneous valid" signal is ON will stop.
- (5) The interlock will not be applied even if the "Manual/Automatic simultaneous valid n-th axis" signal is turned ON for the virtual axis in virtual axis interpolation.
- (6) If the "Manual/Automatic simultaneous valid n-th axis" signal is turned ON for an axis moving with the automatic operation's G90 modal or G53 command and the manual/automatic simultaneous valid axis is moved manually, when the "Manual/Automatic simultaneous valid n-th axis" signal turns OFF, the end point of automatic operation will be deviated by the amount the axis was moved manually. During the G90 modal, the command path will be returned to during the next block.
- (7) If the "Manual/Automatic simultaneous valid n-th axis" signal is turned ON for an axis during reference position return and the axis is manually moved, the reference position return operation will be executed again.

When the reference position is reached manually, the interlock will be released, and the block will end. Manual operation after the block is completed will follow the manual operation block.

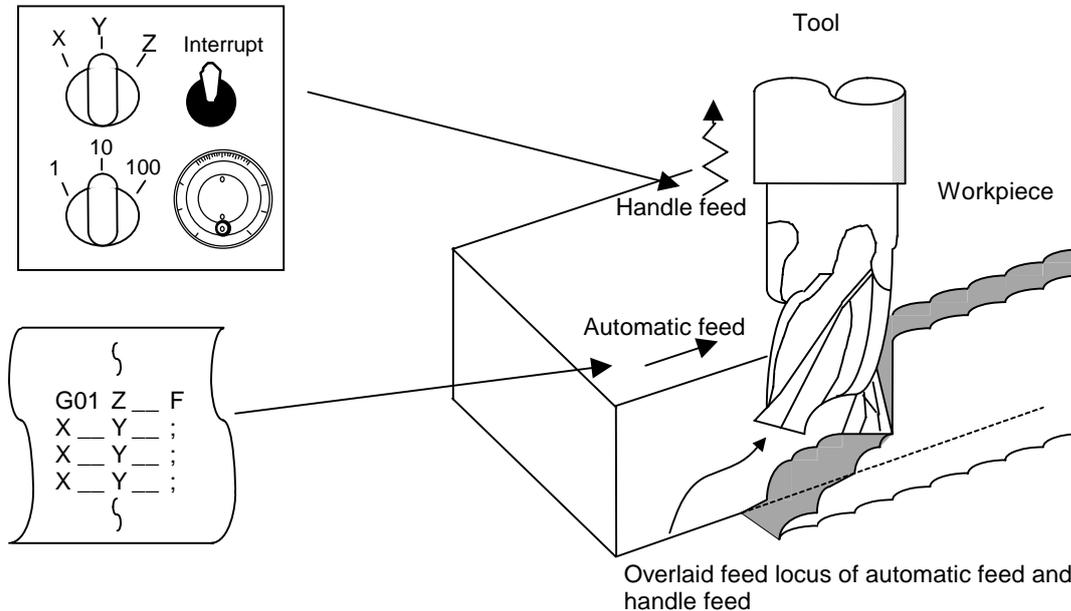
6.16.5 Other Precautions

- (1) If only the automatic operation mode is selected and the manual operation mode is not selected, the "Manual/Automatic simultaneous valid n-th axis" signal will be invalid.
- (2) Even if one automatic operation mode and one manual operation mode are selected, all manual operations of an axis for which the "Manual/Automatic simultaneous valid n-th axis" signal is OFF will be invalid. Note that if the automatic operation handle interrupt option is added, interrupt can be applied with handle feed.
- (3) If the "Manual/Automatic simultaneous valid n-th axis" signal is ON for the 1st and 2nd axes and the "Circular feed in manual mode valid" signal is turned ON during automatic operation, the manual arc feed will not be validated.
If automatic operation is attempted while manual arc feed is valid, the manual arc feed will be invalidated even if the "Manual/Automatic simultaneous valid n-th axis" signal for the 1st and 2nd axes is ON. In this case, none of the axes can be moved with manual operation until the "Circular feed in manual mode valid" signal is turned OFF.
- (4) The "Manual/Automatic simultaneous valid n-th axis" signal is invalid for the chopping axis during chopping.
- (5) The "Manual/Automatic simultaneous valid n-th axis" signal is invalid for the slave axis during synchronous control. When the "Manual/Automatic simultaneous valid n-th axis" signal is turned ON for the master axis, the slave axis will become the manual/automatic simultaneous valid axis.
- (6) Manual random feed will not be executed if even one axis other than the manual/automatic simultaneous valid axis is set as the manual random feed axis. Note that only the manual/automatic simultaneous valid axis is executed during non-interpolation.
- (7) When the manual mode is selected for the manual/automatic simultaneous valid axis while moving in the automatic high-accuracy mode, if the axis is moved manually before the automatic operation decelerates to a stop with the interlock, the speed may change in steps when the manual movement is started. Start manual operation after the automatic operation has decelerated to a stop.

6.17 Handle Interruption

6.17.1 Outline

Section 6.14 explains automatic handle interruption, which enables the operator to interrupt movement using the manual handle in automatic modes (tape, memory, MDI).



6.17.2 Interruptible Conditions

- (1) The automatic handle interrupt function allows you to interrupt the program manually by selecting the manual handle mode in automatic mode selection (tape, memory, MDI, etc.). However the interrupt cannot be executed by the manual handle when an automatic reference position return command (G28, G29, G30), the thread cutting (G33), or the skip command (G31) has been executed or when tapping in the tapping cycle.
- (2) If automatic operation mode such as tape, memory, or MDI is being selected even when an automatic operation pause (including a block stop) is established, automatic handle interruption is enabled.
- (3) If the axis is moved during dwell (G04) command processing by using automatic handle interruption, the dwell count operation will stop. A check is made for the completion of the axis movement, then the dwell count operation continues.
- (4) Automatic handle interruption is enabled even if automatic machine lock has been set. If manual machine lock has been set, the machine does not move; it only updates the POSITION display. If manual machine lock has not been set, the machine moves by the interruption distance by the manual handle and the POSITION display is updated.
- (5) This function is disabled for an axis to which the interlock signal has been input or an axis of which interruption direction is the soft limit.

6.17.3 Interruption Effective Axis

- (1) Automatic handle interruption is enabled only for axes to which manual handle axis selection has been input.
- (2) Automatic handle interruption is enabled for a maximum of three axes. (The number of axes is restricted by the number of handles.)

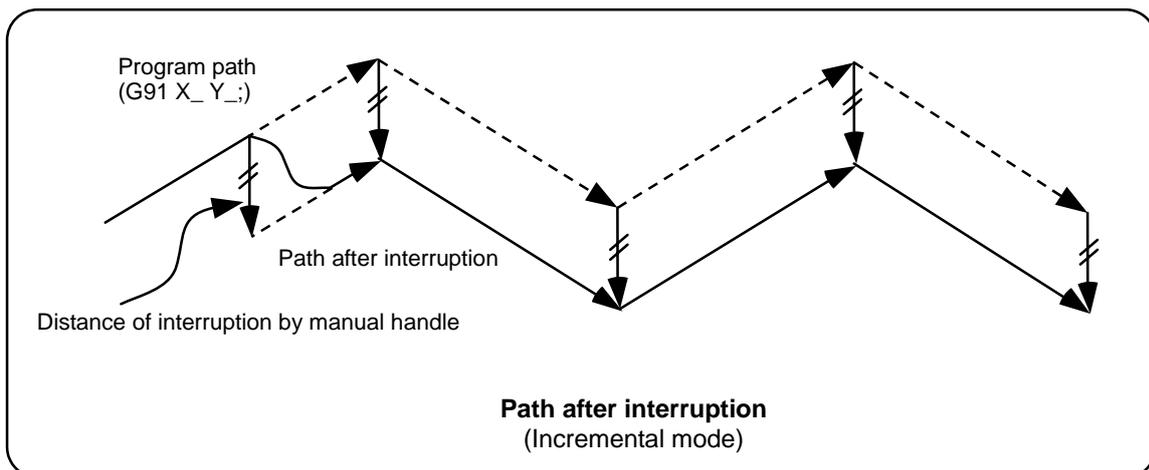
6.17.4 Axis Movement Speed Resulting From Interruption

- (1) The movement speed (Automatic-start movement speed + Speed resulting from manual handle interruption.) of the axis for which handle interruption is executed, may exceed the rapid traverse feed rate during rapid traverse feed command (G00) processing in automatic start. To prevent this, clamp the axis.
- (2) The movement speed (Automatic start movement speed + Speed resulting from manual handle interruption.) of the axis for which handle interruption is executed, may exceed the cutting feed speed during cutting feed command (G01, G02, G03) processing in automatic start. To prevent this, clamp the axis.
- (3) If, during automatic start, manual handle interruption is executed in the same direction for the axis that is moving at an external decelerating speed, the axis movement speed (Automatic start movement speed + Speed resulting from manual handle interruption.) may exceed the external decelerating speed. To prevent this, clamp the axis.
- (4) If an attempt is made to execute interruption at a speed exceeding the clamp speed, the reading on the handle scale does not match the distance of interruption.
- (5) The handle scale factor depends on the selected input of the manual handle/step scale factor.

6.17.5 Path Resulting After Handle Interruption

(1) For incremental (G91) mode

The locus deviates from the program path by the distance of interruption. (See the figure below.)



(2) For absolute (G90) mode

If program absolute position update by the distance of handle interruption is disabled, the locus deviates from the program path by the distance of interruption.

If this update is enabled, the locus returns to the program path during processing of the following command:

For single block running - Return command is issued in the block next to the one for which the interruption has been completed.

For continuous running - Return command is issued in the third block, if the block for which the interruption has been completed is the first block.

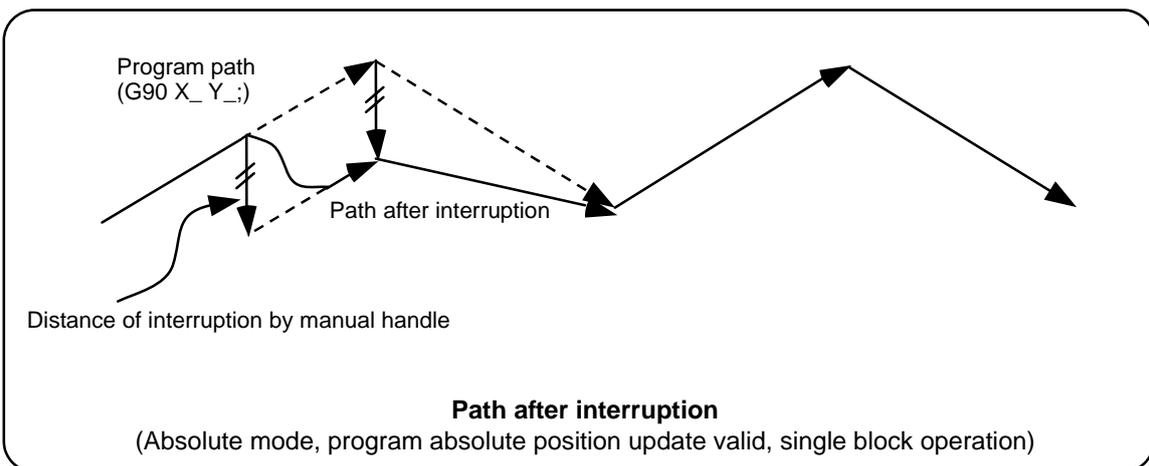
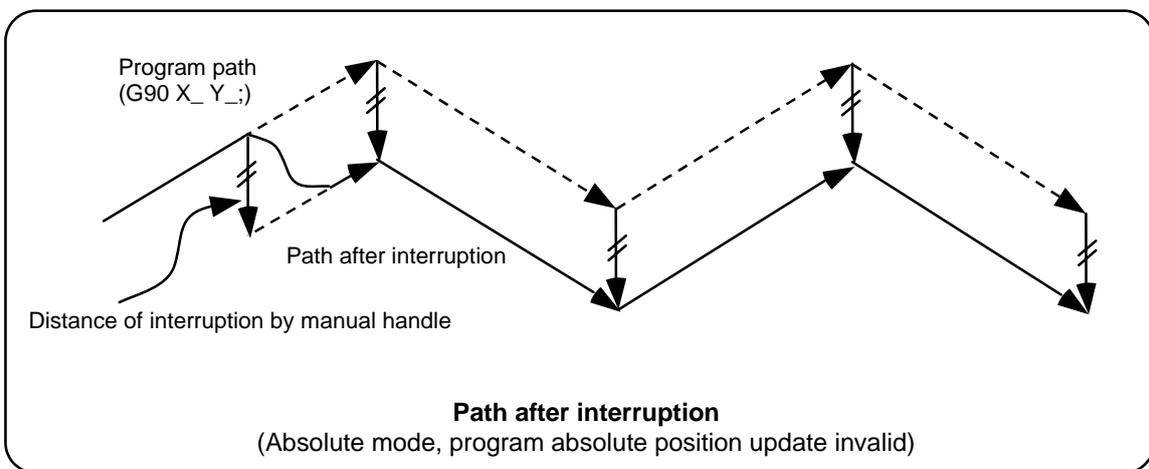
(Note) Both POSITION and MACHINE display include the distance of handle interruption.

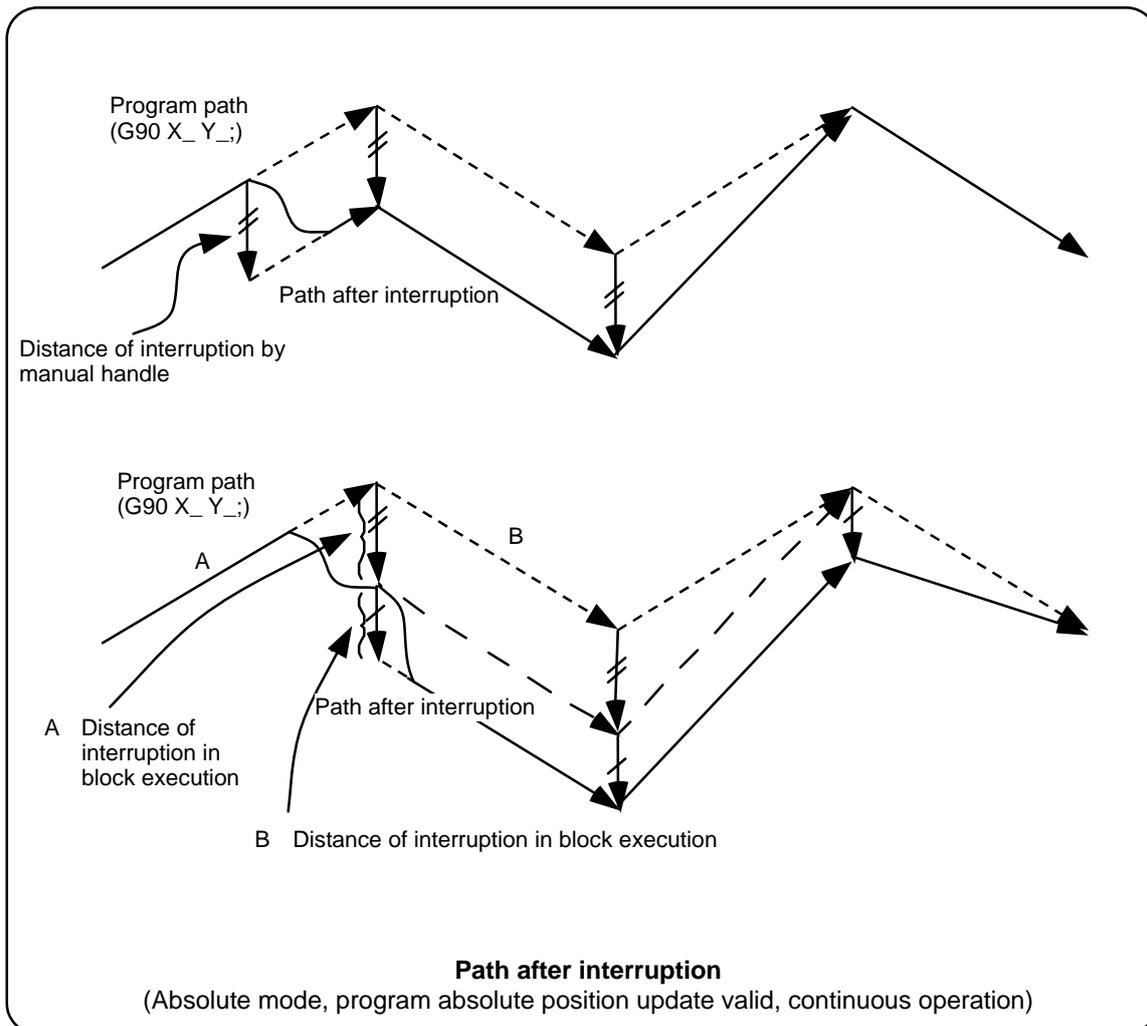
For automatic handle interruption, select whether absolute mode is updated or not, as follows:

- (a) Using machine parameters, set whether the manual absolute changeover switch or parameters are used.
- (b) If the parameters are used, set whether or not absolute position data is updated for each axis, by using machine parameters other than those in item.
- (c) If the manual absolute changeover switch is used, use the machine operation panel switch for selection.

Absolute position update conditions for automatic handle interruption

			Absolute data update		Drive monitor screen manual interruption distance display
Parameter #1145 I_abs	ON "1"	Parameter "#1061 intabs" (Every axis)	ON "1"	Absolute position is updated.	Not updated.
			OFF "0"	Absolute position is not updated.	Updated.
	OFF "0"	PLC interface manual absolute switching	ON	Absolute position is updated.	Not updated.
			OFF	Absolute position is not updated.	Updated.





6.17.6 Handle Interruption in Tool Radius Compensation

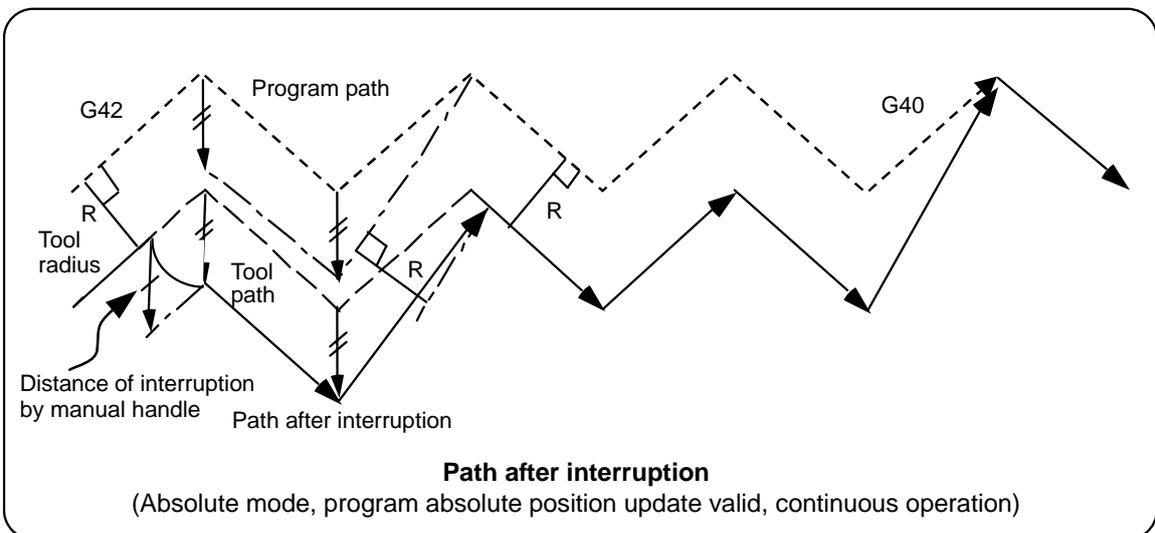
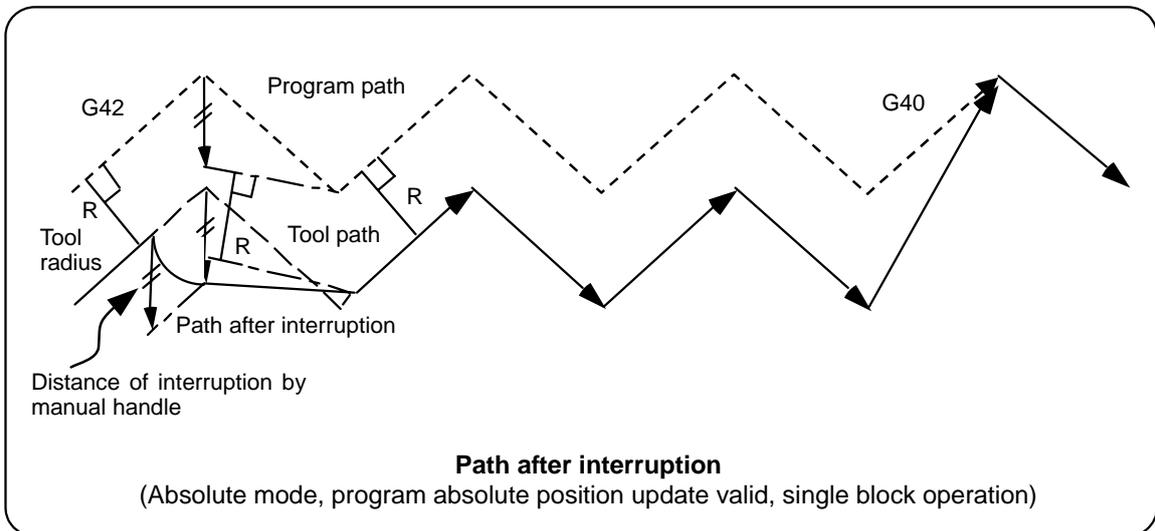
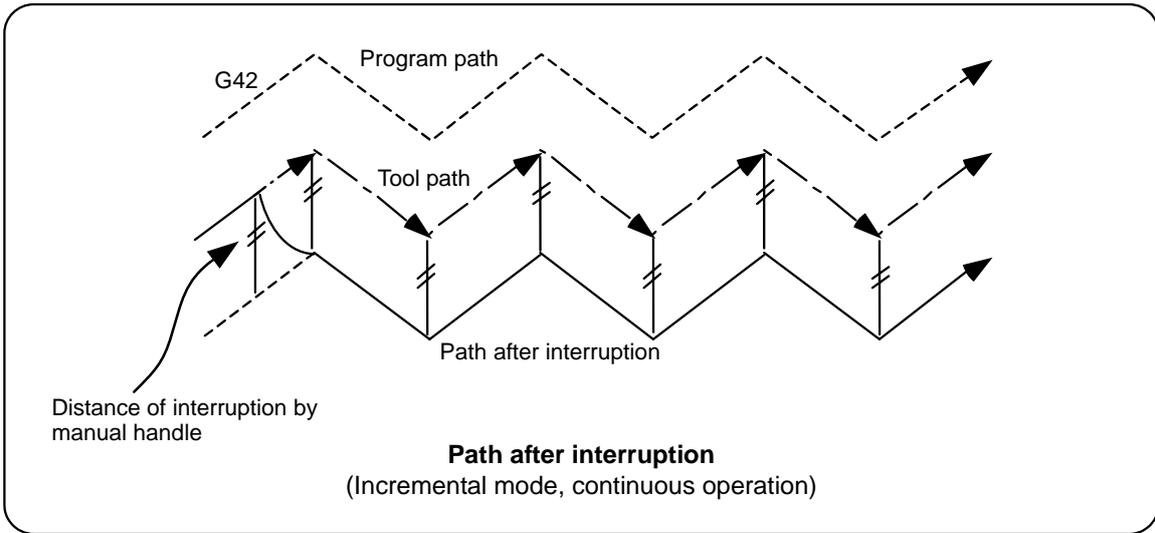
Special movement described below relates only to the tool radius compensation plane axis. It has no influence on the other axes.

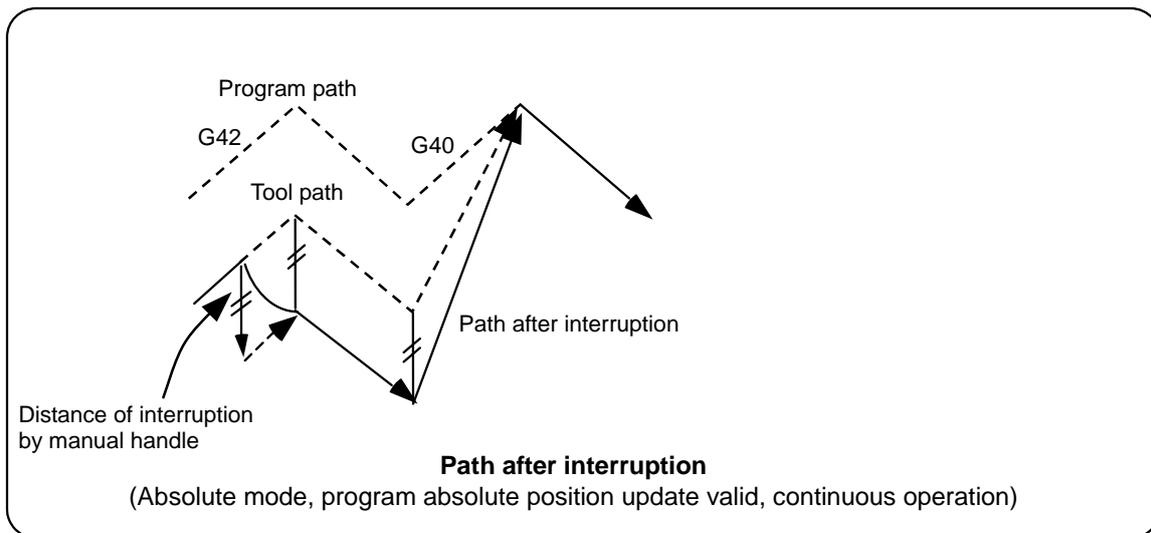
At time of tool radius compensation (G41, G42):

In incremental mode (G91) - The quantity of deviation equals the distance of interruption.

In absolute mode (G90) - If handle interruption is executed in the block for which tool radius compensation (G41, G42) is being executed, the proper tool path will return in the succeeding block. This rule applies only when the program absolute position update is valid during single block running. If program absolute position update is valid during continuous running, the proper tool path will return at the fourth block after the one where the interruption has been completed.

If there is no G block existed between the block where the interruption has been completed and the block that contains the tool radius competition cancel command (G40), the proper tool path will return at the block next to the tool radius compensation cancel command (G40).





6.17.7 Interrupt Amount Reset

Interrupt amount is reset when

- (1) Dog reference position return is executed;
- (2) Emergency stop is released;
- (3) Reset rewind or reset 2 is executed; or
- (4) Reset 1 is executed when the setup parameter "#1151 rstint" is ON.

6.17.8 Operation Procedure

An operation example is given on the assumption that auto operation of XYZ axes is executed in the memory operation mode and the Z axis is used as a handle interrupt axis.

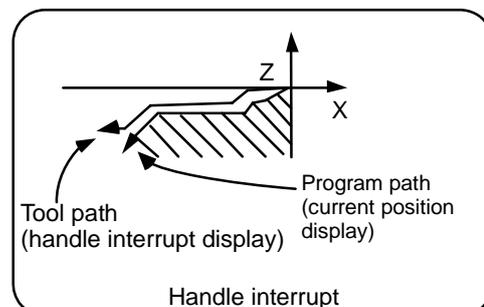
Perform automatic operation.

Turn ON the HANDLE INTERRUPT switch on the machine operation panel.

Select interrupt axis by setting the HANDLE AXIS SELECTION switch on the machine operation panel.

Select the travel distance per handle graduation by setting the HANDLE/ INCREMENTAL MAGNIFICATION switch.

Turn the manual handle in the + or - direction and change the Z axis cut depth amount.



When M02 or M30 is executed, automatic operation will stop.

6.18 All Axis Machine Lock

- (1) When the "ALL AXES MACHINE LOCK" switch is turned ON, the NC commands for the manual operation or automatic operation movement can be executed without moving the machine.
The current position display on the setting and display unit will be counted.
- (2) The automatic operation speed during machine lock can be selected as the commanded speed or machine lock speed using the parameters.
 - Commanded speed - The movement is executed at the feedrate commanded in the program.
Thus, the time is the same as actual machining.
 - Machine lock speed - The movement commands are processed at the rapid traverse rate, and the dwell time, etc., is ignored. Thus, the program check, etc., can be carried out faster.
- (3) If the "ALL AXES MACHINE LOCK" switch is changed during automatic operation, the automatic operation will stop after the block currently being executed is completed, and then the setting will be validated.
- (4) During reference position return (G28, G30), the movement will be controlled with the machine lock status up to the middle point. The machine lock status will be ignored from the middle point to the reference position.
- (5) If the "MACHINE LOCK" switch is changed during manual operation, the setting will be validated after the feed is stopped once.
- (6) The M, S, T and B commands are executed according to the program.
- (7) When the "MACHINE LOCK" is turned OFF after the axis is moved in the "MACHINE LOCK" ON state, the current position display and the machine position will not match.
If AUTO START is pressed in this state, the difference between the current position and the machine position will be added to the movement amount.
If RESET is pressed, the current position display will be changed to match the machine position. Thus, after turning "MACHINE LOCK" OFF, press RESET before starting operation.

6.19 Each Axis Machine Lock

By turning ON the "MACHINE LOCK" switch provided for each axis, machine lock will be applied only on the control axis (axes) turned ON. All matters, other than the machine lock for each axis, are the same as all axes machine lock.

6.20 Deceleration Check

6.20.1 Functions

The purpose of the deceleration check is to reduce the machine shock that occurs when the control axis feedrate is suddenly changed, and prevent corner roundness. The check is carried out at block joints.

(1) Deceleration check during rapid traverse

The deceleration check is always carried out at the block joints (before executing the next block) during rapid traverse.

(2) Deceleration check during cutting feed

The deceleration check is carried out at the block joints (before executing the next block) during cutting feed when any one of the following conditions is valid.

- (a) When the error detect switch is ON.
- (b) When G09 (exact stop check) is commanded in the same block.
- (c) When G61 (exact stop check mode) has been selected.

(3) Designating deceleration check

The deceleration check by designating a parameter includes "deceleration check specification type 1" and "deceleration check specification type 2". The setting is selected with the parameter "#1306 InpsTyp".

- (a) Deceleration check specification type 1 ("#1306 InpsTyp" = 0)

The G0 and G1 deceleration check method can be selected with the base specification parameter deceleration check method 1 "#1193 inpos" and deceleration check method 2 "#1223 aux07/bit1".

Parameter	Rapid traverse command	Parameter	Other than rapid traverse command (G1 : other than G0 command)	
#1193 Inpos	G0→XX (G0+G9→XX)	#1223 AUX07 (BIT-1)	G1+G9→XX	G1→XX
0	Command deceleration check	0	Command deceleration check	No deceleration check
1	In-position check	1	In-position check	

(Note 1) XX expresses all commands.

(Note 2) "#1223 aux07" is the part system common parameter.

- (b) Deceleration check specification type 2 ("#1306 InpsTyp" = 1)

Rapid traverse and cutting feed in-position are designated with the parameter "#1193 inpos".

Parameter	Command block		
	G0	G1+G9	G1
0	Command deceleration check	Command deceleration check	No deceleration check
1	In-position check	In-position check	No deceleration check

(Note 1) "#1193 inpos" is the parameter per part system.

(Note 2) "G0" means the rapid traverse, and "G1" means the cutting feed.

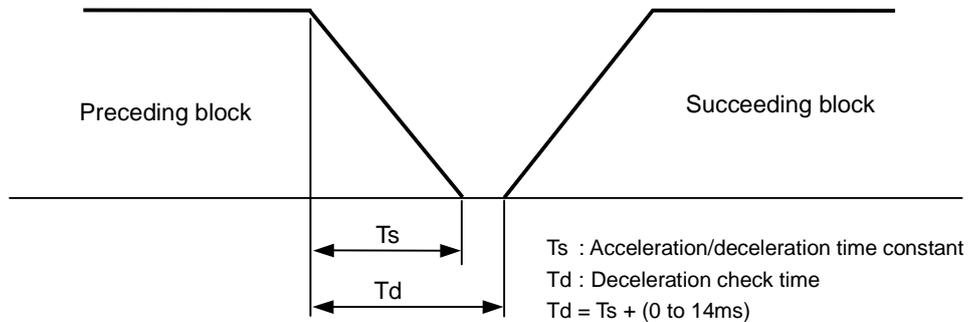
6.20.2 Deceleration Check Method

(1) Command deceleration check

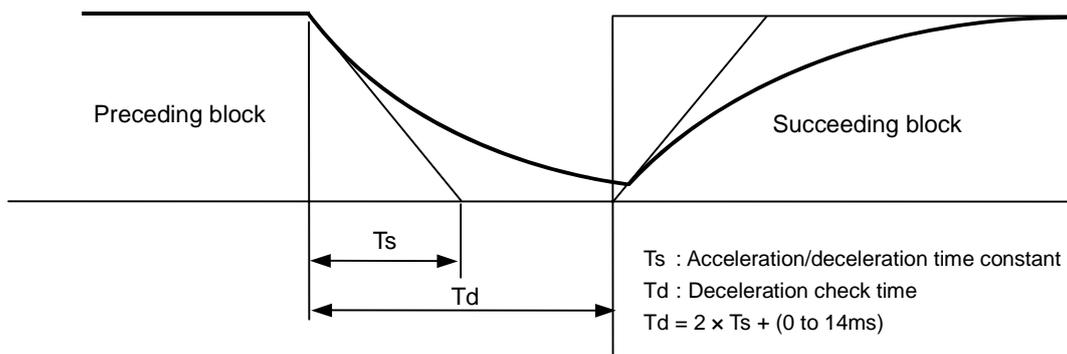
After interpolation for one block has been completed, the completion of the command system deceleration is confirmed before execution of the next block.

The time required for the deceleration check is determined according to the acceleration/deceleration mode and acceleration/deceleration time constant.

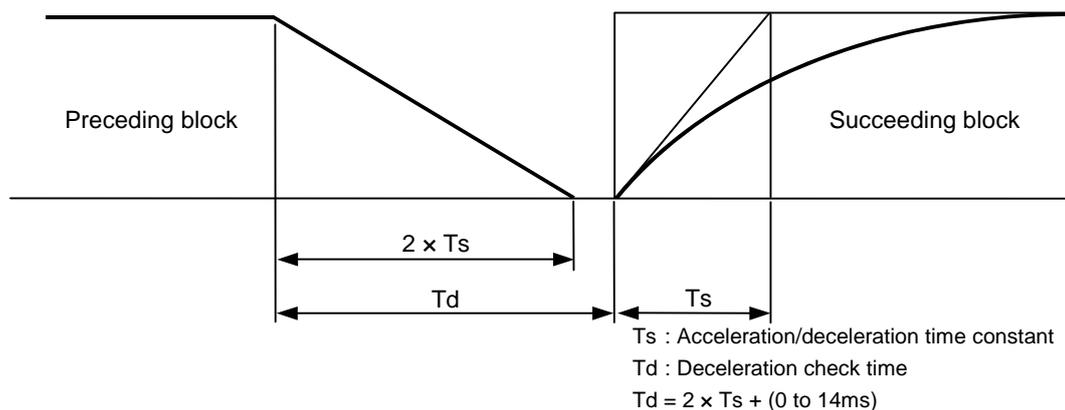
(a) For linear acceleration/deceleration



(b) For exponential acceleration/deceleration



(c) For exponential acceleration and linear deceleration



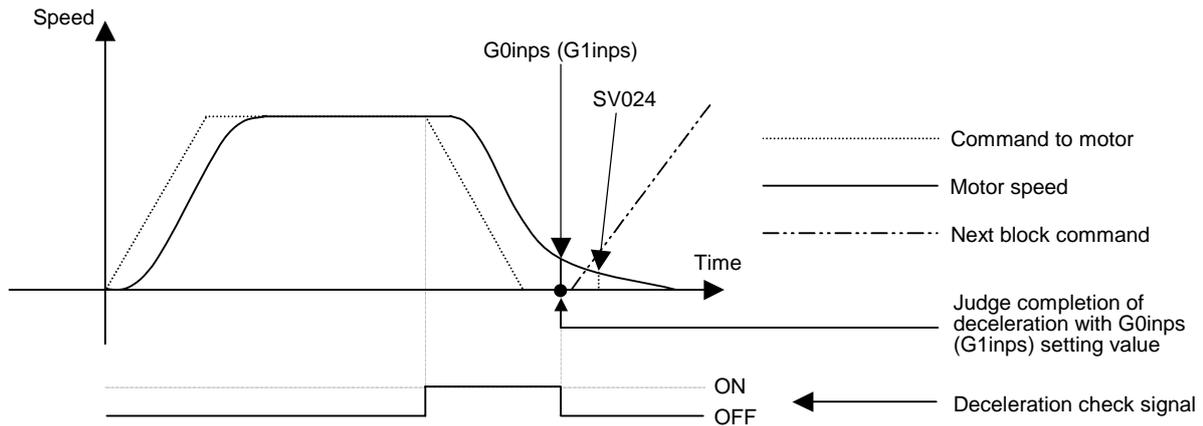
The deceleration check time required during rapid traverse is the longest rapid traverse deceleration check time of all axes. This check time is determined by the rapid traverse acceleration/deceleration mode and rapid traverse acceleration/deceleration time constant of simultaneously commanded axes.

The deceleration check time required during cutting feed is determined in the same manner. It is the longest cutting feed deceleration check time of all axes. This check time is determined by the cutting feed acceleration/deceleration mode and cutting feed acceleration/deceleration time constant of simultaneously commanded axes.

(2) In-position check

When the in-position check is valid, the command deceleration check is carried out. After that, it is confirmed that the servo system positional error is less than the parameter setting value, and the next block is executed.

The in-position check width can be designated with the servo parameter in-position width (SV024). Note that G0 and G1 can be designated independently with the axis specification parameter G0 in-position check width (G0inps) and G1 in-position check width (G1inps). If both the servo parameter and axis specification parameter are set, the larger value will have the priority.



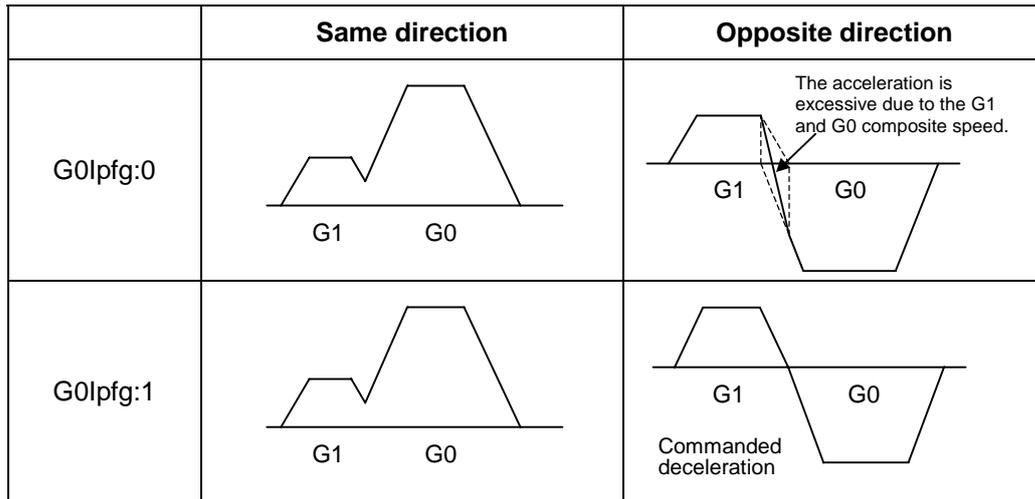
If the SV024 setting value is larger, in-position check will end when the SV024 setting value is established.

6.20.3 Deceleration Check When Opposite Direction Movement Is Reversed

Deceleration check cannot be designated for G1 → G0 or G1 → G1, but it can be designated in the following manner only when the movement reverses to the opposite direction in successive blocks. Deceleration check can also be executed if even one axis is moving in the opposite direction while several axes are interpolating.

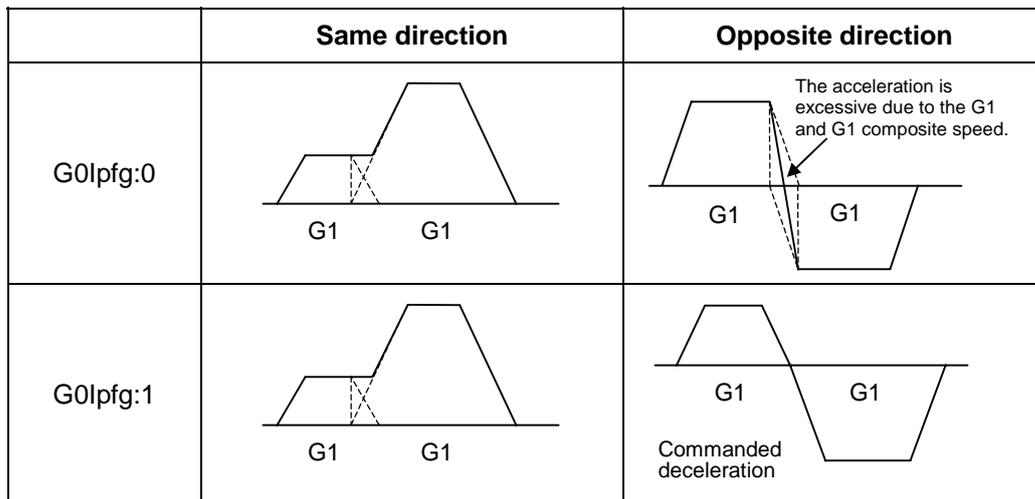
(1) Designating deceleration check for G1 → G0 opposite direction movement reversal

If the axis movement reverses to the opposite direction in a G1 to G0 successive block, the deceleration check for the movement in the opposite direction can be changed with the base specification parameter G1 → G0 deceleration check (#1502 G0lpfg).



(2) Designating deceleration check for G1 → G1 opposite direction movement reversal

If the axis movement reverses to the opposite direction in a G1 to G1 successive block, the deceleration check for the movement in the opposite direction can be changed with the base specification parameter G1 → G1 deceleration check (#1503 G1lpfg).



6.20.4 Parameters

(1) Designation of deceleration check

Base specification parameters

#	Items	Details	Setting range	
1193	inpos	The definitions are changed with the setting of "#1306 InpsTyp Deceleration check specification type".		
		When Deceleration check method 1 is selected	Select the deceleration check method for G0. 0: Command deceleration check 1: In-position check	0/1
		When Deceleration check method 2 is selected	Select the deceleration confirmation method for the positioning or cutting command. 0: G0, G1+G9 Command deceleration check 1: G0, G1+G9 In-position check	0/1
1223	aux07 (bit1)	Deceleration check method 2	Select the deceleration check method in G1+G9. 0: Command deceleration check in G1+G9 1: In-position check in G1+G9 The deceleration check is not performed for the commands except G1+G9. When "#1306 InpsTyp deceleration check specification type" is set to "1" (Deceleration check specification type 2), this parameter will be invalid.	0/1
1306	InpsTyp	Deceleration check specification type	Select the parameter specification type for the G0 or G1 deceleration check. 0: Deceleration check specification type 1 G0 is specified with "#1193 inpos", and G1+G9 with "#1223 aux07/bit1". 1: Deceleration check specification type 2 G0 or G1+G9 is specified with "#1193 inpos".	0/1

(2) Deceleration check during opposite direction travel

Base specification parameters

#	Items	Details	Setting range	
1502	G0lpfg	G1 → G0 deceleration check	Select whether to perform a deceleration check when the travel direction is changed from G1 to G0. 0: Not perform 1: Perform	0/1
1503	G1lpfg	G1 → G1 deceleration check	Select whether to perform a deceleration check when the travel direction is changed from G1 to G1. 0: Not perform 1: Perform	0/1

(3) Designation of in-position check width

(a) Servo parameter

#	Items	Details	Setting range	
2224	SV024 INP	In-position detection width	Set the in-position detection width. Set the accuracy required for the machine. The lower the setting is, the higher the positioning accuracy gets; however, the cycle time (setting time) becomes longer. The standard setting value is "50".	0 - 32767 (μm)

(b) Axis specification parameter

#	Items	Details	Setting range	
2077	G0inps	G0 in-position width	Set the in-position width for G0. Between SV024 and this parameter, the parameter with a larger value will be applied. When "0" is set, this parameter will be invalid: only SV024 will be available.	0.000 to 99.999 (mm)
2078	G1inps	G1 in-position width	Set the in-position width for G1. Between SV024 and this parameter, the parameter with a larger value will be applied. When "0" is set, this parameter will be invalid: only SV024 will be available.	0.000 to 99.999 (mm)

6.20.5 Precautions**(1) Designating deceleration check**

- When in-position check is valid, set the in-position width in the servo parameters.

(2) Deceleration check for opposite direction movement reversal

- When deceleration check is valid (G0lpfg=1), deceleration check will be executed when the axis reverses its movement to the opposite direction at the G1 → G0 successive block regardless of whether G0 non-interpolation is ON or OFF.
- When deceleration check is valid (G0lpfg=1), deceleration check will be executed when the axis reverses its movement to the opposite direction at the G1 → G0 successive block even in the fixed cycle.
- In the G1 → G28, G1 → G29 or G1 → G30 successive blocks, deceleration check will always be executed when the G1 movement is completed, when movement to the intermediate point is completed and when movement to the return point is completed. Note that if the base specification parameter's simple zero point return "#1222 aux06/bit7" is valid, the base specification parameter G1 → G0 deceleration check (G0lpfg) will be followed when the G1 movement is completed and when movement to the intermediate point is completed. (Deceleration check will always be executed when movement to the return point is completed even in this case.)

(3) Designating in-position width

- The in-position width (programmable in-position check width) designated in the machining program has a priority over the in-position width set in the parameters (SV024, G0inps, G1inps).
- When error detect is ON, in-position check will be forcibly carried out.

(4) Deceleration check in G1 → G0, G1 → G1 opposite direction movement reversal during high-speed machining mode

- When the axis movement reverses to the opposite direction in a G1 → G1 successive block during the high-speed machining mode, the commanded deceleration will not take place even if G1lpfg is set to 1. Note that the G0lpfg setting will be followed if the axis direction reverses to the opposite direction in a G1 → G0 successive block.

(5) Deceleration check in movement including spindle/C-axis

- The deceleration check for spindle/C-axis movement command is as described in the table below. That is because a vibration and so on occurs in the machine when the position loop gain (#13002 PGN) is changed during the axis movement.

Parameter	Rapid traverse command	Parameter	Other than rapid traverse command (G1 : other than G0 command)	
#1193 Inpos	G0→XX (G0+G9→XX)	#1223 AUX07 (BIT-1)	G1→G0 (G1+G9→XX)	G1→G1
0	Command deceleration check	0	In-position check (Applicable only to SV024)	No deceleration check
1	In-position check	1		

(Note 1) When G1 command is issued, the in-position check is performed regardless of the deceleration check parameter.

(Note 2) XX expresses all commands.

(6) Deceleration check in polar coordinate interpolation / milling interpolation / cylindrical start / cancel command

- The deceleration check in polar coordinate interpolation / milling interpolation / cylindrical start / cancel command are as follows.

Parameter: #1223 aux07 BIT1	Deceleration check method
0	Command deceleration check
1	In-position check

6.21 Tool Return and Escape

6.21.1 Outline

Due to tool breakage or temporary monitoring, the tool is manually escaped after the machine program is interrupted by feed hold. This function allows the machine to approach the interruption point via the transit point that was designated as the automatic start point and restart the machining, after the workpiece is checked or the tool is changed.

Up to two transit points can be designated in the escape path.

After the tool is changed, the compensation amount cannot be changed.

6.21.2 Retracting Method

Designating the interruption point (return position)

In order to designate a halted point, turn ON a transit point switch when operation is stopped by feed hold or single block. When recognition of the halted point is completed, this signal turns ON and the tool return and escape mode will be established.

Escape methods

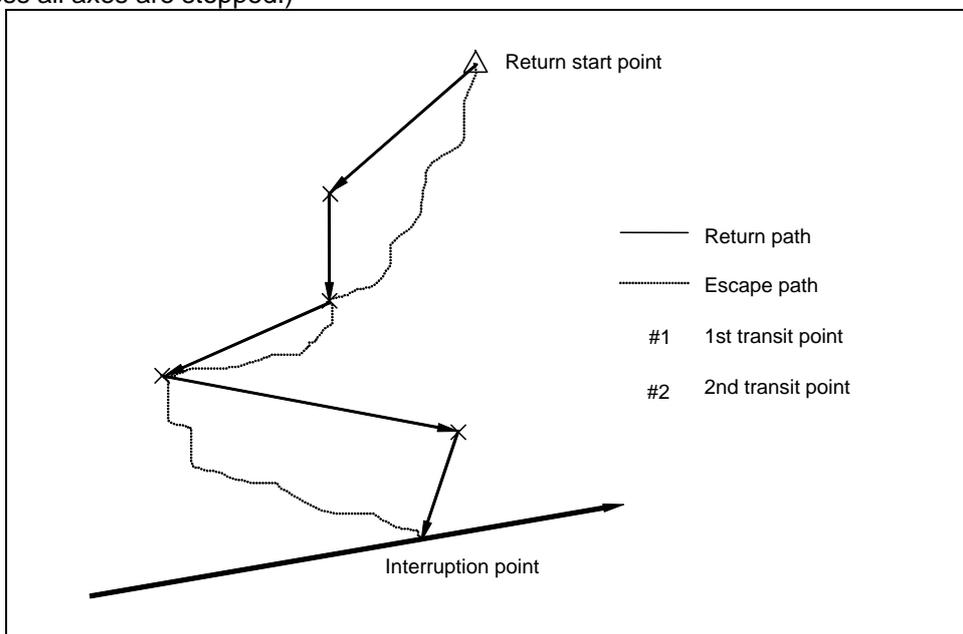
Escape is carried out manually by changing to the manual mode (handle, JOG, JOG rapid traverse). At this time, the transit point for designating the return path can be designated.

Escape is also possible by automatic operation handle interrupt.

Designating the transit point

Two transit points can be designated. To designate, move to the transit point with the manual mode, and then turn ON the transit point switch to memorize it.

The transit point 1 (#1) is the point designated first after interruption, and the transit point 2 (#2) is the point designated last. Stop all axes before designating the transit points. (The transit points will not be designated unless all axes are stopped.)



Example of escape operation

- (1) Stop the program by turning "FEED HOLD" switch or "SINGLE BLOCK" switch ON.
- (2) Designate the interruption point by turning the transit point switch ON.
- (3) Move to transit point #1 with the manual mode (JOG, handle, step).
- (4) Turn the transit point switch ON. ➔ Transit point #1 will be memorized.
- (5) Move to transit point #2 with the manual mode (JOG, handle, step).
- (6) Turn the transit point switch ON. ➔ Transit point #2 will be memorized.
- (7) Move to the return start point with the manual mode (JOG, handle, step).
- (8) Change to the automatic operation mode. ➔ The return will start.

Starting the return

To start the return, change from the manual mode to the automatic mode, and move toward the transit point with cycle start. The return and escape mode is entered during the return.

Resetting the escape mode

The escape mode is reset when reset 1 is input, emergency stop occurs the return is completed. Escape is also possible by automatic operation handle interrupt.

Return path

The axis returns in the following order.

Return start point → Transit point #2 → Transit point #1 → Interruption point

When transit point #2 is not designated	When only interruption point is designated without transit point designation
When transit point #2 is not designated, the return start point will be regarded as the transit point #2.	When only the interruption point is designated without a transit point designation, the return start point will be regarded as transit point #1.
Return path: Return start point (= transit point #2) → transit point #1 → interruption point	Return path: Return start point (= transit point #1) → interruption point

After the interruption point is reached, the retract return mode is canceled. Automatic operation starts, and the remaining blocks are executed.

Movement path

3 kinds of movement path are shown below. For types (1) and (3), the return type must be set with the parameters.

(1) Return start point to #2

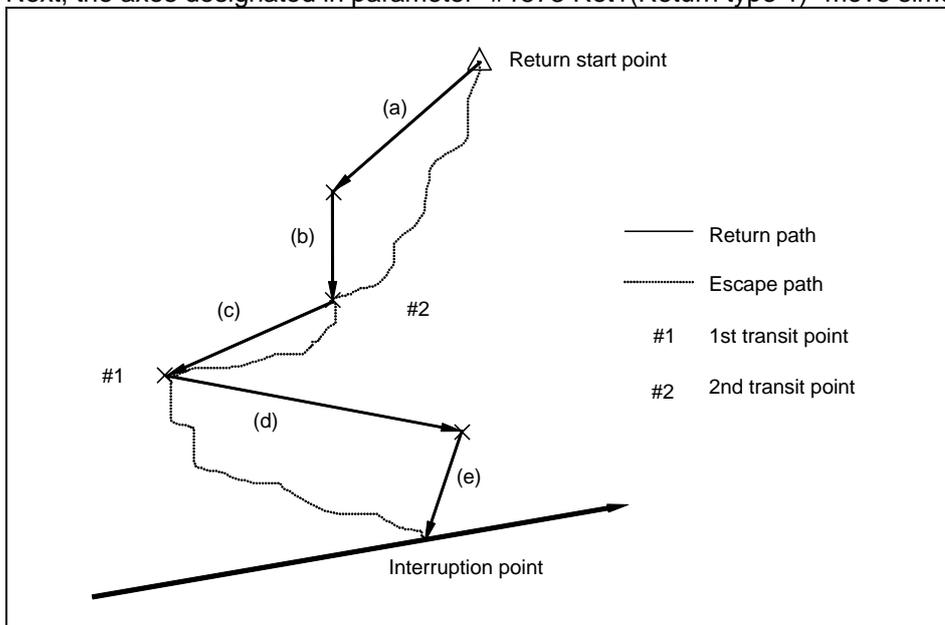
- (a) First, the axes other than that designated in parameter "#1574 Ret1(Return type 2)" simultaneously move.
- (b) Next, the axes designated in parameter "#1574 Ret1(Return type 2)" move simultaneously.

(2) #2 to #1

- (c) All axes move simultaneously.

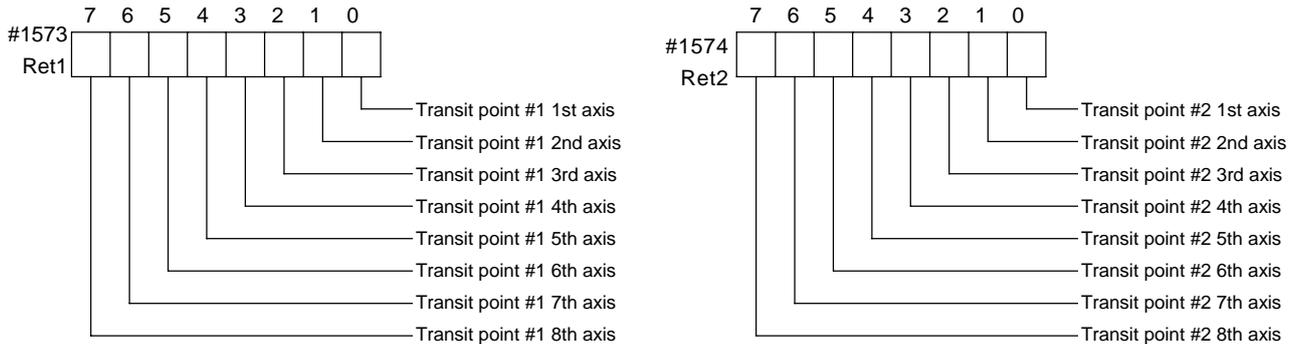
(3) #1 to interruption point

- (d) First, the axes other than that designated in parameter "#1573 Ret1(Return type 1)" simultaneously move.
- (e) Next, the axes designated in parameter "#1573 Ret1(Return type 1)" move simultaneously.



Setting the parameters "#1573 Ret1(Return type 1)" and "#1574 Ret1(Return type 2)"

The return type are set with the parameters "#1573 Ret1(Return type 1)" and "#1574 Ret1(Return type 2)". Set the axis to be moved as a bit unit in each parameter. The axes designated with "#1573 Ret1(Return type 1)" move to the interruption point as shown with (e) in the previous drawing. The non-designated axes move as shown with (d) in the drawing. The axes designated with "#1574 Ret1(Return type 2)" move to the #2 transit point shown as (b) in the drawing, while the non-designated axes move as shown with (a). The parameter details are as shown below.

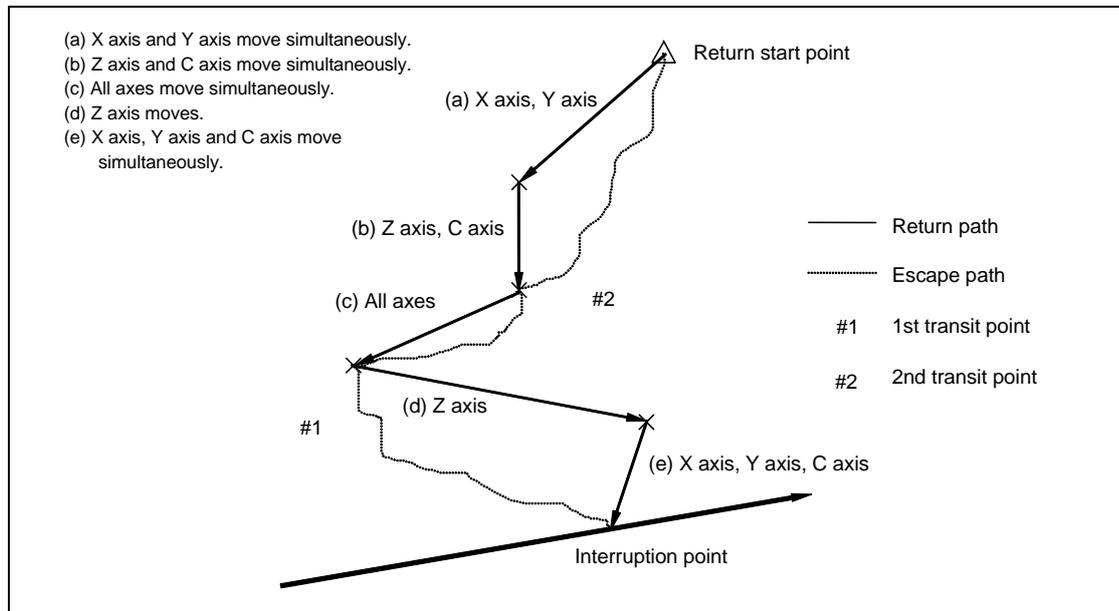


6.21.3 Example of Operation

When using a 4-axis system with the X axis as the 1st axis and, Y, Z, C in the order, set the parameters "#1573 Ret1(Return type 1)" and "#1574 Ret1(Return type 2)" as shown below.

#1573 Ret1 00001011
 #1574 Ret2 00001100

In this case, the return movement path will be as shown below.



6.21.4 Precautions

- (1) If reset is applied between the designation of the interruption point and the restart of machining (in other words, during the escape mode), the memorized transit points and interruption point will be canceled. The halted or stopped automatic operation will be reset and finished.
- (2) The tool compensation amount cannot be changed during the escape mode.
- (3) The manual random feed function cannot be used during the escape mode.
- (4) As for the override, the manual override is valid.
- (5) The single block is valid.
- (6) If reference position return is carried out during the tool return and escape, the tool compensation amount will be invalidated when the reference position return is completed. Thus, if the tool is compensated before tool escape, when the tool is returned after reference position return, the path will shift by the tool compensation amount at the block following the block containing the interruption point. Thus, if the operation mode is changed to reference position return during tool return and escape, an alarm will occur.

Alarm occurrence conditions	<ul style="list-style-type: none"> • If the reference position return mode is selected during the tool return and escape mode, operation alarm "M01 Tool ofs invld after R-pnt (0021)" will occur.
Alarm reset conditions	<ul style="list-style-type: none"> • The error is cleared if the operation mode is changed to other than reference position return before the axis performs reference position return. • The error is cleared when reference position return is completed. • The error is cleared if reset 1 is input or the emergency stop button is pushed.

6.22 External Deceleration

6.22.1 Outline

When the "External deceleration+ n-th axis" (*+EDT1 to *+EDT8), "External deceleration- n-th axis" (*-EDT1 to *-EDT8) signal, an external input from the PLC program, is input, this function immediately drops the feedrate to the external deceleration speed set in the parameters.

6.22.2 Explanation of Function

- (1) The external deceleration signal is provided for each axis and for each movement direction (+/-). The speed will decelerate when a signal in the direction matching the movement direction is input.
- (2) When the axis is returned in the reverse direction, the command speed will be returned to immediately.
- (3) During manual operation or automatic operation's non-interpolation positioning, only the axis for which a signal matching the movement direction is input will decelerate.
- (4) During automatic operation interpolation, if there is any axis for which the input signal matches the movement direction, the feedrate will be lowered to the external deceleration speed.

6.22.3 Combination with Other Functions

- (1) When the following functions are enabled, the external deceleration is disabled.
 - Machine lock axis
 - Synchronous tap modal
 - Tapping mode
 - While moving in the approach speed at automatic reference position return command (G28) or dog-type reference position return during the manual reference position return mode
 - The axis moving with chopping
 - During thread cutting
 - Virtual axis
- (2) If the external deceleration speed is slower than the overridden speed, the speed will drop to the external deceleration speed.
- (3) When calculating the machining time, the external deceleration signal input is not considered.
- (4) When an inch command is issued, the axis moves at the speed obtained by converting the external deceleration speed into inches.
- (5) During the manual speed command, the external deceleration signal is valid for the movement axis as well as the axis and direction that is the same as the selected feed axis (+/-).
- (6) During manual random feed, the external deceleration axis is valid for the same axis and direction as the axis for which manual random feed is valid.
(If the manual random feed is valid for the axis but the movement amount is 0, the external deceleration signal + for the same axis will be valid.)
- (7) During manual arc feed, the external deceleration signal for same axis and direction as the selected feed axis (+/-) will be valid instead of the movement axis.

6.22.4 Precautions

- (1) If the external deceleration signal is input while moving at a speed lower than the external deceleration speed, the feedrate will not be affected.
- (2) Excluding during G00 non-interpolation, the composite speed of each axis will be the external deceleration speed during automatic operation.
- (3) The signal is valid after resetting and applying emergency stop.

6.23 Reference Position Retract

6.23.1 Outline

This function returns immediately to a set reference position when the Reference position retract signal is input. This function is used to return to a set position for changing the tool.

6.23.2 Detailed Description

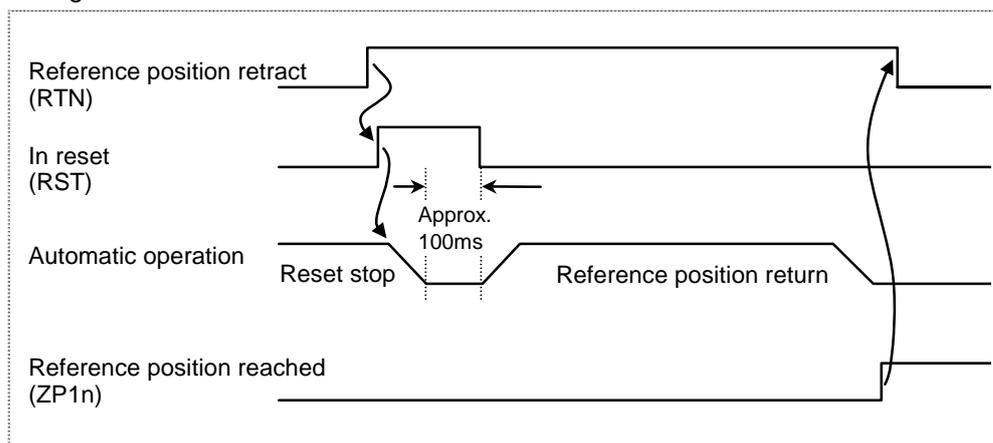
Reference position return is executed when this signal is turned ON. At the rising edge of the signal, the program is automatically reset (reset & rewind) and then reference position return is executed.

During the automatic or MDI operation, the operation is interrupted and stopped by the reset, and reference position return is executed.

If this signal is input during execution of a tap cycle in the automatic or MDI operation modes, the "Tap retract possible" signal will be output by the reset interruption, and the return operation will be the tap retract operation. The tap retract is completed at the initial point, and after that the reference position return will be carried out.

- (1) If there are two or more axes, set the return order with parameter "#2019 revnum".
- (2) When the reference position is reached, the corresponding "Reference position reached" signal will be output.
- (3) This signal must be held until the "Reference position reached" signal is output. If it is turned OFF midway, the return operation will be interrupted and stop. If the signal is input again, the operation will restart from execution of resetting.
- (4) The reference position return speed is handled in the same manner as the normal reference position return speed.
- (5) In this function, an axis returns to the reference position set by the Reference position selection code 1 and 2 signal.
- (6) Even if the return signal is input during the thread cutting cycle, it will be invalid. However, if the return signal is executed in a block other than the thread cutting block, the return operation will be executed.
- (7) The return signal is invalid if the coordinate system is not established. An alarm "M01 Ref point retract invalid" will occur when the return signal is input.

<Timing chart>



6.24 Spindle Orientation

6.24.1 Outline

(1) Orientation

This function stops the spindle rotation at a certain position when using the digital spindle. When the orientation command is used, the spindle will rotate several times and then stop at the orientation point.

The orientation position differs depending on the detector.

When the encoder orientation (PLG and external encoder/ring sensor) is used:

At the Z-phase position

When the magnetic sensor (proximity switch) is used:

At the magnetic sensor installation position

(2) Multi-point orientation

This function performs orientation to a position other than the Z-phase position by inputting a shift amount with the parameter or PLC. The shift amount is 0 to 35999. (Unit: $360^\circ/36000=0.01^\circ$)

(Note 1) Multi-point orientation cannot be executed when using the magnetic sensor.

(Note 2) Orientation is possible only when the gear ratio is 1:1 for the PLG orientation.

(The orientation is completed at the PLG encoder's Z-phase, so when using reduction gears, the orientation points will be generated at several points during one spindle rotation.)

(3) Orientation imposition advance output

This function turns the spindle in-position signal ON as soon as the spindle reaches within the second in-position width (parameter "#3132 ori_inp2" (2nd in-position width for orientation)). Then, the spindle 2nd in-position signal is turned ON as soon as the spindle reaches within the in-position width (parameter "#13024 SP024 INP" (In-position width)).

Since orientation completion can be predicted using this function, it is possible to eliminate the sequence delay time, etc. for tool changes and other such operations, thereby achieving a faster tact time.

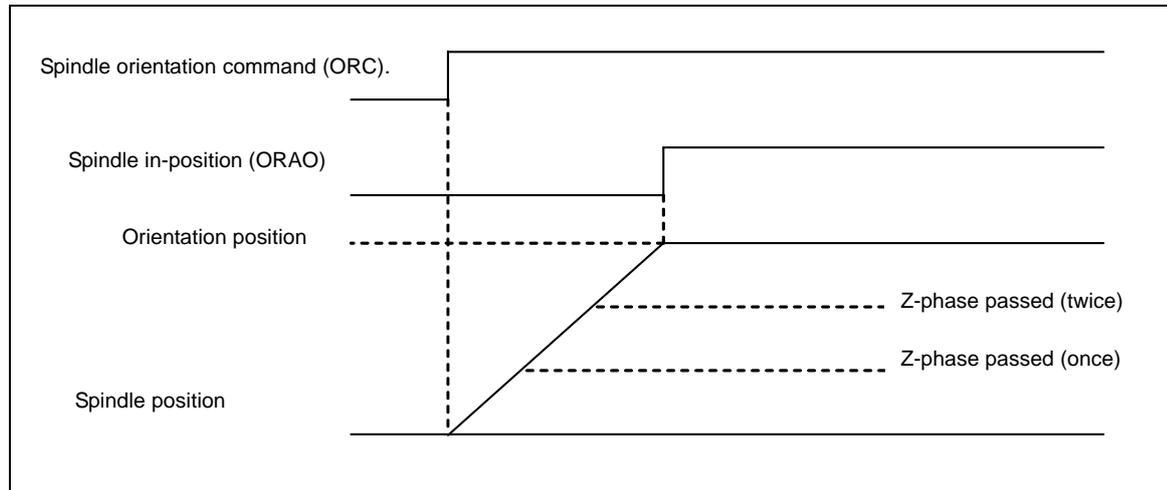
6.24.2 Orientation Operation

When Z-phase has not yet been passed, pass Z-phase twice (excluding reciprocations within one rotation), and then position to the orientation position.

When Z-phase has been passed already, immediately position to the orientation position.

The spindle does not rotate if Z-phase has been passed and the orientation position is already established when orientation command is issued.

When the spindle has been rotating in the direction opposite from that of orientation rotation when the orientation command is issued, orientation operation will be executed after decelerating to stop.



The Acceleration/Deceleration Patterns

(1) The acceleration/deceleration patterns at orientation/position loop reference position return

For the acceleration/deceleration patterns at orientation/position loop reference position return (C axis, tapping), the time constant is set by the parameter "#3115 sp2_t1" to "#3118 sp2_t4" (Time constant in orientation/position loop reference position return). The orientation acceleration/deceleration time is calculated with the following expression.

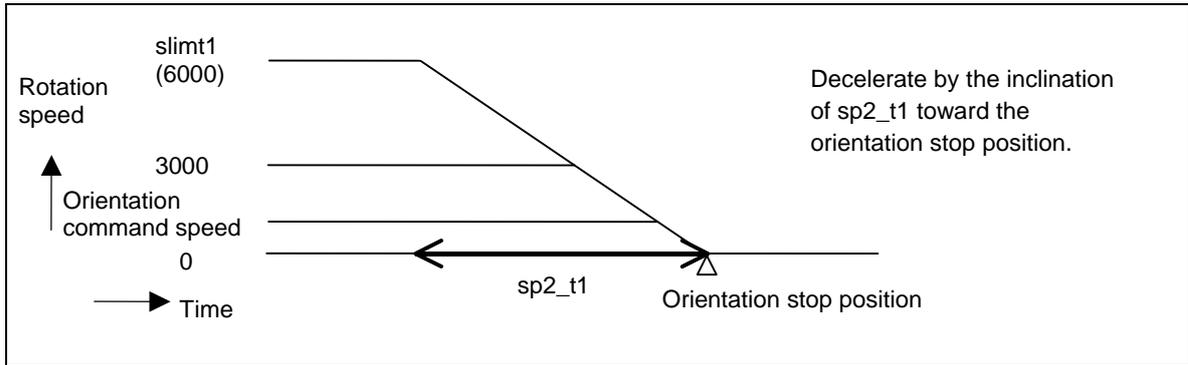
(Example: The acceleration/deceleration time for gear 00)

$$\text{The acceleration/deceleration time} = \text{sp2_t1} \times \frac{\text{Orientation command speed (\#3107 ori_spd)}}{\text{Limit rotation speed (\#3001 slimt 1)}}$$

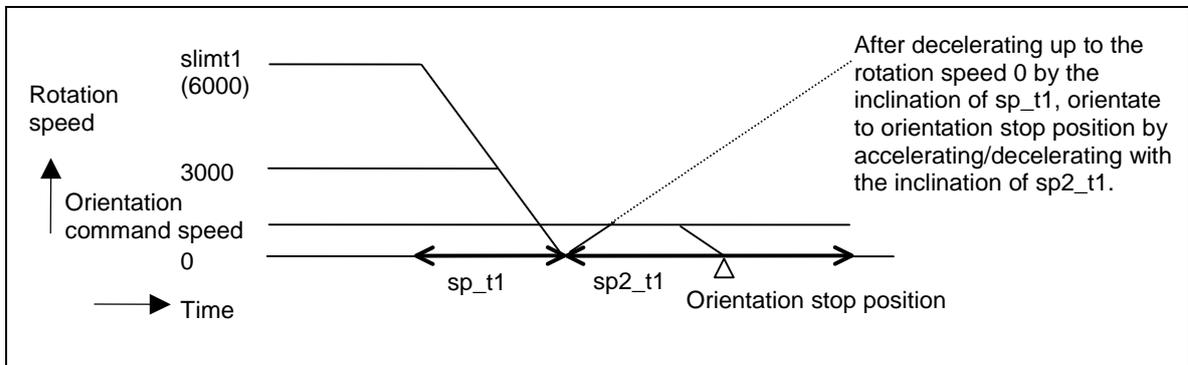
For position loop reference position return, the parameter "#3110 tap_spd" (Synchronous tapping zero point return speed) or "#3112 cax_spd" (Spindle C axis zero point return speed) is used for position loop reference position return (C axis, tapping) instead of the orientation command speed.

In any case, when "#3101 sp_t 1" is larger than "#3102 sp_t 2" (Time constant for spindle rotation with S command), "#3101 sp_t 1" is used.

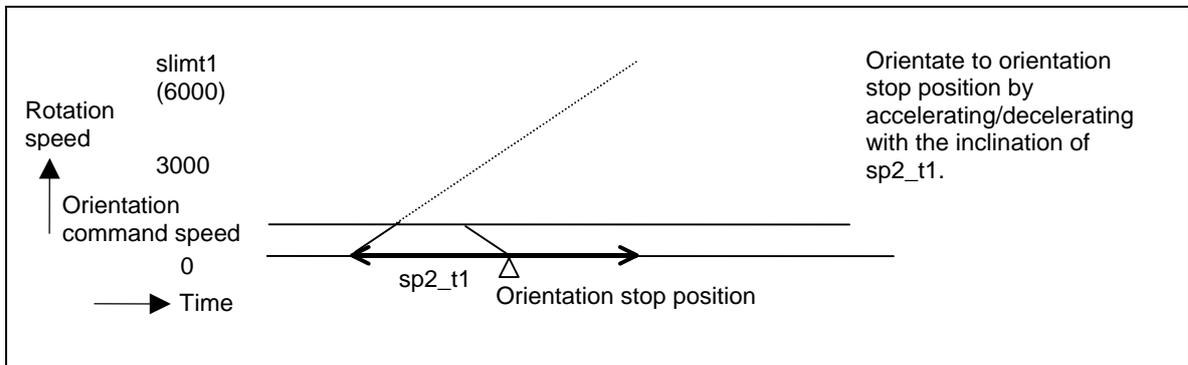
(a) During spindle rotation, and the orientation direction is the same



(b) During spindle rotation, and the orientation direction is opposite

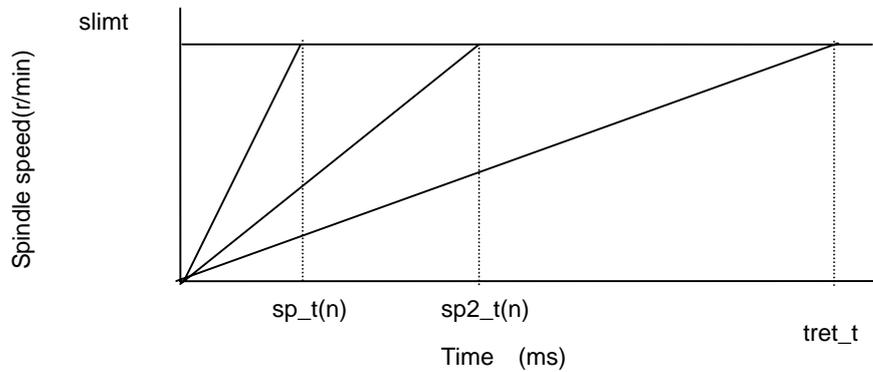


(c) When the spindle is stopping



(2) Time constant

The parameter "#3115 sp2_t1" to "#3118 sp2_t4" (Time constant in orientation/position loop reference position return) are used as the time constant of orientation/position loop reference position return. The parameter "#3124 tret_t" (Turret indexing time constant) is used at the turret indexing. Set the relation to the parameters "#3101 sp_t 1" to "#3104 sp_t 4" (Time constant for spindle rotation with S command) as follows.

**Gain**

If the orientation is commanded when the spindle has been stopping, usually the spindle rotates to orientation position using the position loop gain set in the parameter "#13002 SP002(PGN)" (Position loop gain interpolation mode).

By setting the parameter "#3106 zrn_typ/bitE" (Interpolation mode selection in orientation), if the orientation is commanded when the spindle has been stopping, the spindle rotates to orientation position using the parameter "#13001 SP001(PGV)" (Position loop gain Non-interpolation mode). When the parameter "#3106 zrn_typ/bitF" (Spindle zero point detection with contactless switch) or the parameter "#3121 tret" (Turret indexing) is set, the parameter "#3106 zrn_typ/bitE" (Interpolation mode selection in orientation) setting is not enabled.

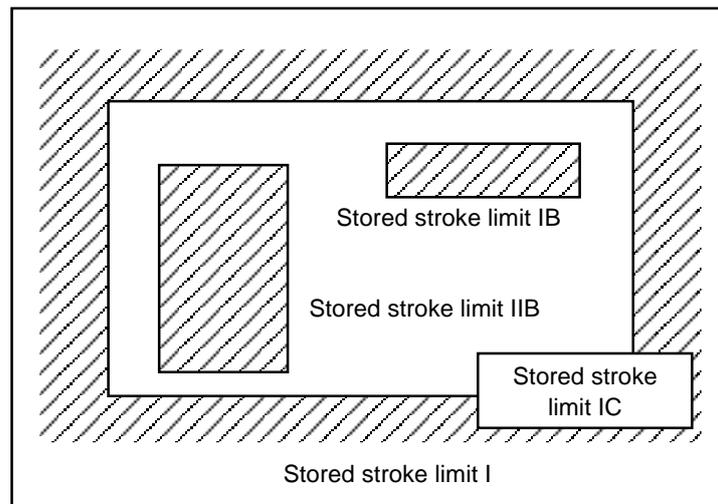
7. Other Functions

7.1 Stored Stroke Limit

Outline

Three tool entry prohibited ranges can be set with stored stroke limit I, stored stroke limit II, IIB and stored stroke limit IB. Part of the prohibited range on the outside of soft limit I can be validated with stored stroke limit IC.

The stored stroke limit II and IIB select the outer side (II) and inner side (IIB) of the entry prohibited range with parameters.



If the tool tries to move over the set range, an alarm will appear, and the axis will decelerate to a stop.

If the prohibited range is entered and an alarm occurs, movement will be possible only in the direction opposite the entry direction.

[Valid Conditions of Stored Stroke Limit]

When using the relative position detection system, the stored stroke limit is invalid until the reference position return is completed after the power is turned ON.

The stored stroke limit can be validated even if the reference position return is not yet completed, by setting parameter #2049 type = 9.

If the absolute position detection is valid when using the absolute position detection system, the stored stroke limit will be validated immediately after the power is turned ON.

[Stored stroke limit coordinates]

The stored stroke limit check is carried out in the basic machine coordinate system established by the reference position return. To validate the stored stroke limit even when the reference position return is not yet completed, check the stored stroke limit using the temporary basic machine coordinate system (basic coordinate system defined when the power was previously turned OFF).

When the 1st dog-type reference position return is completed after the power is turned OFF, the correct coordinate system is established.



CAUTION



Always set the stored stroke limit. Failure to set this could result in collision with the machine end.

(Note) The axis movement possible when the reference position return has not yet completed is limited to manual and handle feed only. Automatic operation is validated after the reference position return is completed.

Detailed explanation

The stored stroke limit sets a prohibited range with the parameters or program command. The maximum and minimum values of the prohibited range are set as coordinate values (radius values) on the basic machine coordinate system for each axis. If the parameters (#8204 OT-CHECK-N and #8205 OT-CHECK-P), for setting the prohibited range, are set to the same value, the stroke will not be checked. This function is valid only for the axis for which the reference position is established.

If the machine enters the prohibited range, an operation error will occur, and the machine movement will stop. The alarm can be reset by moving the erroneous axis in the opposite direction.

During automatic operation, if an alarm occurs with even one axis, all axes will decelerate to a stop. During manual operation, only the axis that caused the alarm will decelerate to a stop. The axis will always stop at a position before the prohibited range. The distance between the prohibited range and stop position will depend on the feedrate.

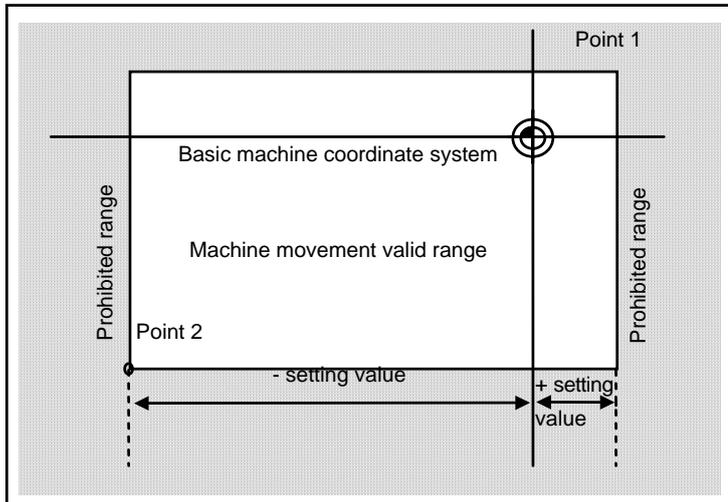
The stored stroke limits I, II, IIB, IB and IC are handled as follows.

Type	Prohibited range	Explanation		Range setting parameter	Validating conditions
I	Outside	<ul style="list-style-type: none"> Set by the machine maker. When used with II, the narrow range designated by the two types becomes the movement valid range. 		#2013 OT- #2014 OT+	<ul style="list-style-type: none"> Zero point return is completed. #2013 and #2014 are not set to the same value.
II	Outside	<ul style="list-style-type: none"> Set by the user. Select II or IIB with the parameters. 	<ul style="list-style-type: none"> #8210 OT-INSIDE:0 Used with I. 	#8204 OT-CHECK-N #8205 OT-CHECK-P	<ul style="list-style-type: none"> Zero point return is completed. #8204 and #8205 are not set to the same value. #8202 OT-CHECK OFF: 0
IIB	Inside		<ul style="list-style-type: none"> #8210 OT-INSIDE: 1 		
IB	Inside	<ul style="list-style-type: none"> Set by the machine maker. 		#2061 OT_1B- #2062 OT_1B+	<ul style="list-style-type: none"> Zero point return is completed. #2061 and #2062 are not set to the same value.
IC	Outside	<ul style="list-style-type: none"> Set by the machine tool builder. 		#2061 OT_1B- #2062 OT_1B+	<ul style="list-style-type: none"> #2061 and #2062 are not set to the same value. #2063 OT_1C type: 2

7.1.1 Stored Stroke Limit I

This is a stroke limit function used by the machine maker. The boundary is set with the parameters (axis specification parameters "#2013 OT-" and "#2014 OT+"). The outside of the set boundary is the prohibited range.

When used with the stored stroke limit II function, the narrow range designated by the two types becomes the movement valid range.



The following values are set with the coordinate values in the basic machine coordinate system.

Point 1: #2014 OT+

Point 2: #2013 OT-

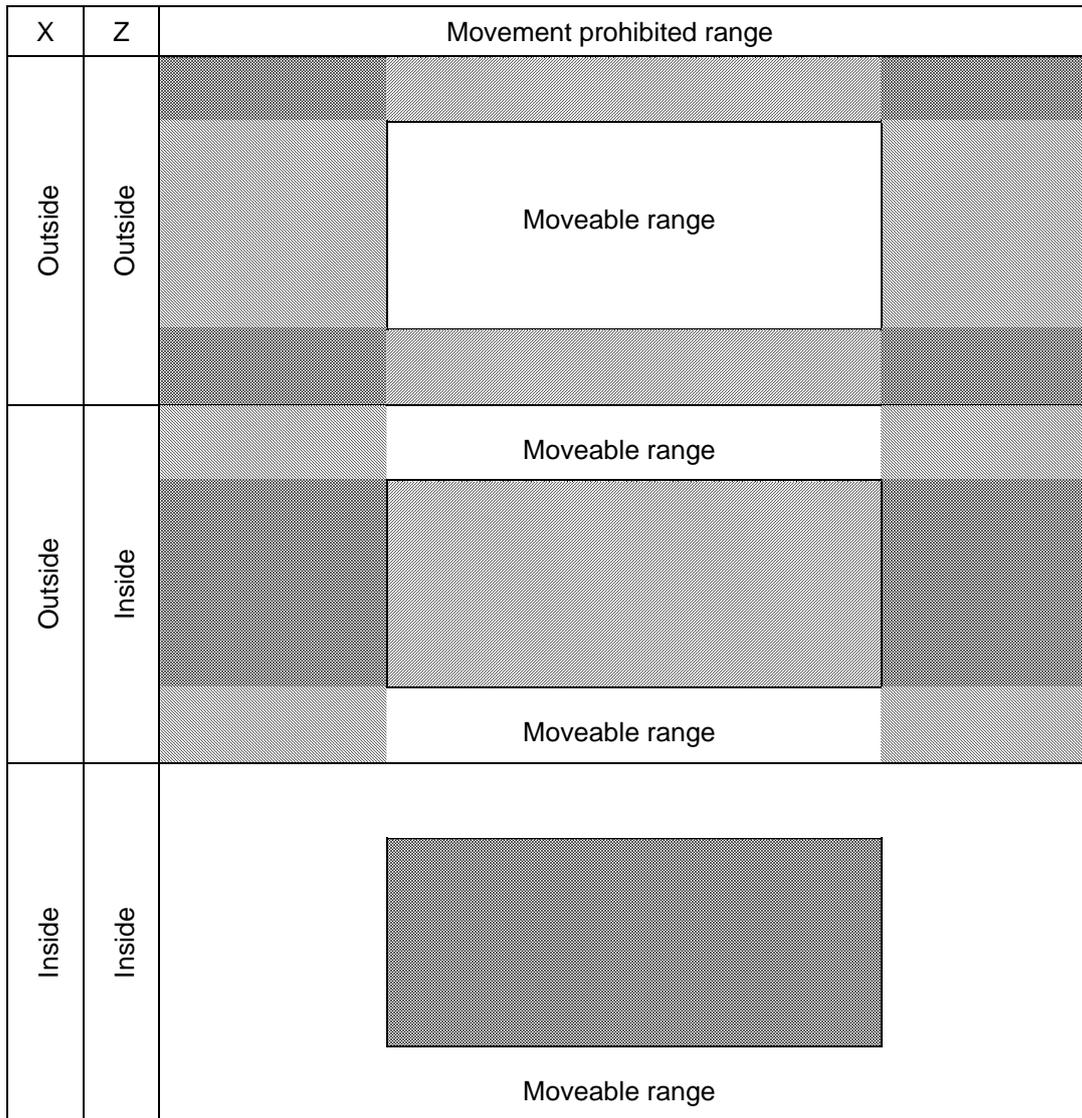
7.1.2 Stored Stroke Limit II

The boundary is set with the parameters (axis parameters #8204 OT-CHECK-N, #8205 OT-CHECK-P) or with the program commands. Either the inside or the outside of the set boundary is the prohibited range. Whether the inside or outside of the range is prohibited is determined by parameter (#8210 OT-INSIDE). When the inside is selected, this function is called stored stroke limit IIB.

When using program commands, entry of the tool into the prohibited range is prohibited with G22, and entry into the prohibited range is enabled with G23.

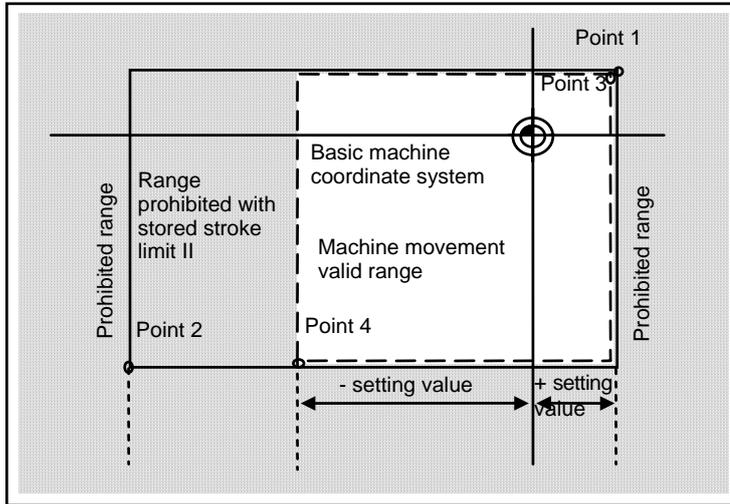
The stored stroke limit II function can be invalidated for each axis with the parameter setting (#8202 OT-CHECK OFF: 1).

"#8210 OT-INSIDE" can be set for each axis, but when used in combination, the following type of operation is possible.



Stored stroke limit II (When prohibited range is on outside)

When used with the stored stroke limit I function, the narrow range designated by the two types becomes the movement valid range.



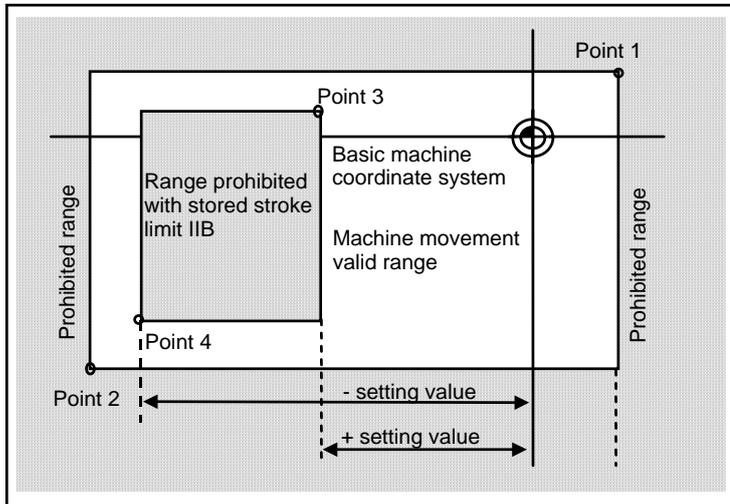
The following values are set with the coordinate values in the basic machine coordinate system.

- Point 3 and 4:
- #8205 OT-CHECK-P
- #8204 OT-CHECK-N

Points 1 and 2 are the prohibited range set with stored stroke limit I.

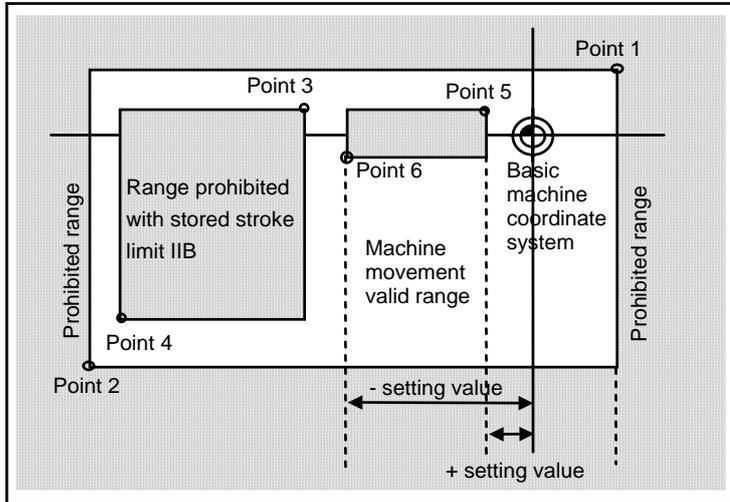
Stored stroke limit IIB (When prohibited range is on inside)

A range except for that of the stored stroke limit I becomes the movement prohibited range.



7.1.3 Stored Stroke Limit IB

The boundary is set for each axis with the parameters (axis parameters "#2061 OT_1B-" and "#2062 OT_1B+"). The inside of the set boundary is the prohibited range.



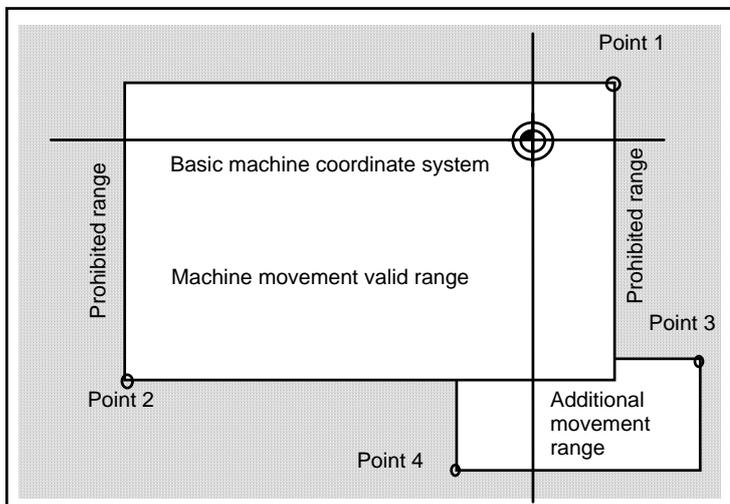
The following values are set with the coordinate values in the basic machine coordinate system.

- Point 3 : #2062 OT_1B+
- Point 4 : #2061 OT_1B-

Points 1 and 2 are the prohibited range set with stored stroke limit I, and points 3 and 4 are the prohibited range set with stored stroke limit II.

7.1.4 Stored Stroke Limit IC

The boundary is set for each axis with the parameters (axis parameters "#2061 OT_1B-" and "#2062 OT_1B+"). The inside of the set boundary is the machine movement valid range. This is valid when the axis parameter #2063 is set to 2, and cannot be used with soft limit IB.



The following values are set with the coordinate values in the basic machine coordinate system.

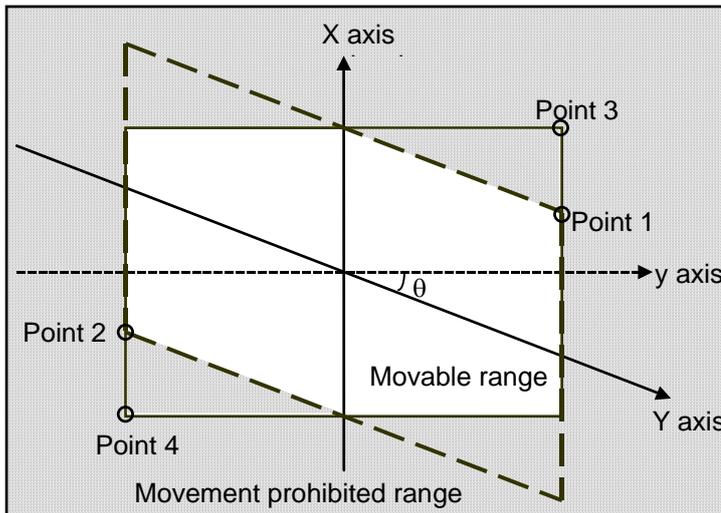
- Point 3 : #2062 OT_1B+
- Point 4 : #2061 OT_1B-

Points 1 and 2 are the prohibited range set with stored stroke limit I.

7.1.5 Movable Range during Inclined Axis Control

By setting #2063 OT_1Btype to "3", the inclined axis control axis can be checked with the program coordinates using the stored stroke limit IB/IC range setting (#2061, #2062). The stored stroke limit IB and IC cannot be used together at this time.

By using this function with stored stroke limit I, the check can be carried out simultaneously with the actual axis and program coordinate value. In this case, the range that does not fit into either of the following two prohibited ranges will be the movable range.



The following values are set with the coordinate values in the basic machine coordinate system.

Point 3: #2062 OT_1B+
Point 4: #2061 OT_1B-

Points 1 and 2 are the prohibited range set with stored stroke limit I.

7.1.6 Stored Stroke Limit for Rotation Axis

Stored stroke limits I and II are used as the stored stroke limit for the rotation axis. The area between the maximum and minimum values of the prohibited range's parameters, which does not contain the 0 point of the basic machine coordinate system, is the entry prohibited range.

The prohibited range parameters for the rotation axis can be set to establish "maximum value < minimum value". This will be handled in the same manner as if "maximum value > minimum value".

(Example)

Stored stroke limit I (maximum value and minimum value of prohibited range parameter)

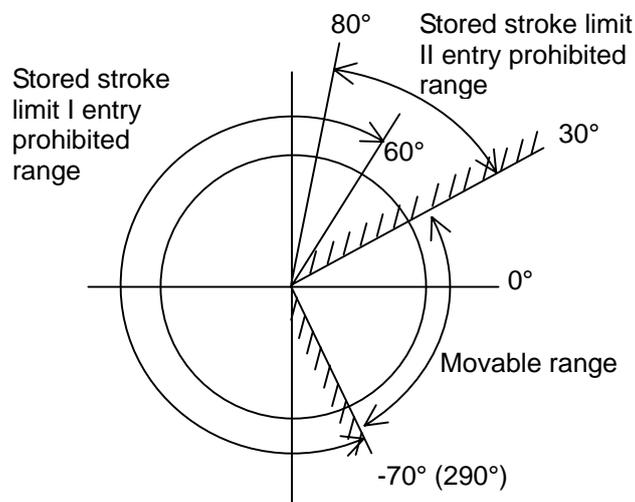
#2013 OT- : -70.000°

#2014 OT+ : 60.000°

Stored stroke limit II (maximum value and minimum value of prohibited range parameter)

#8204 Soft limit - : 30.000°

#8205 Soft limit + : 80.000°



(Note) Do not use stored stroke limits IB, IIB or IC. Invalidate stored stroke limits IB, IIB and IC by setting the parameters as shown below.

#8210 OT INSIDE: 0 (stored stroke limit II valid, IIB invalid)

#2061, #2062 set to same value (stored stroke limits IB and IC invalid)

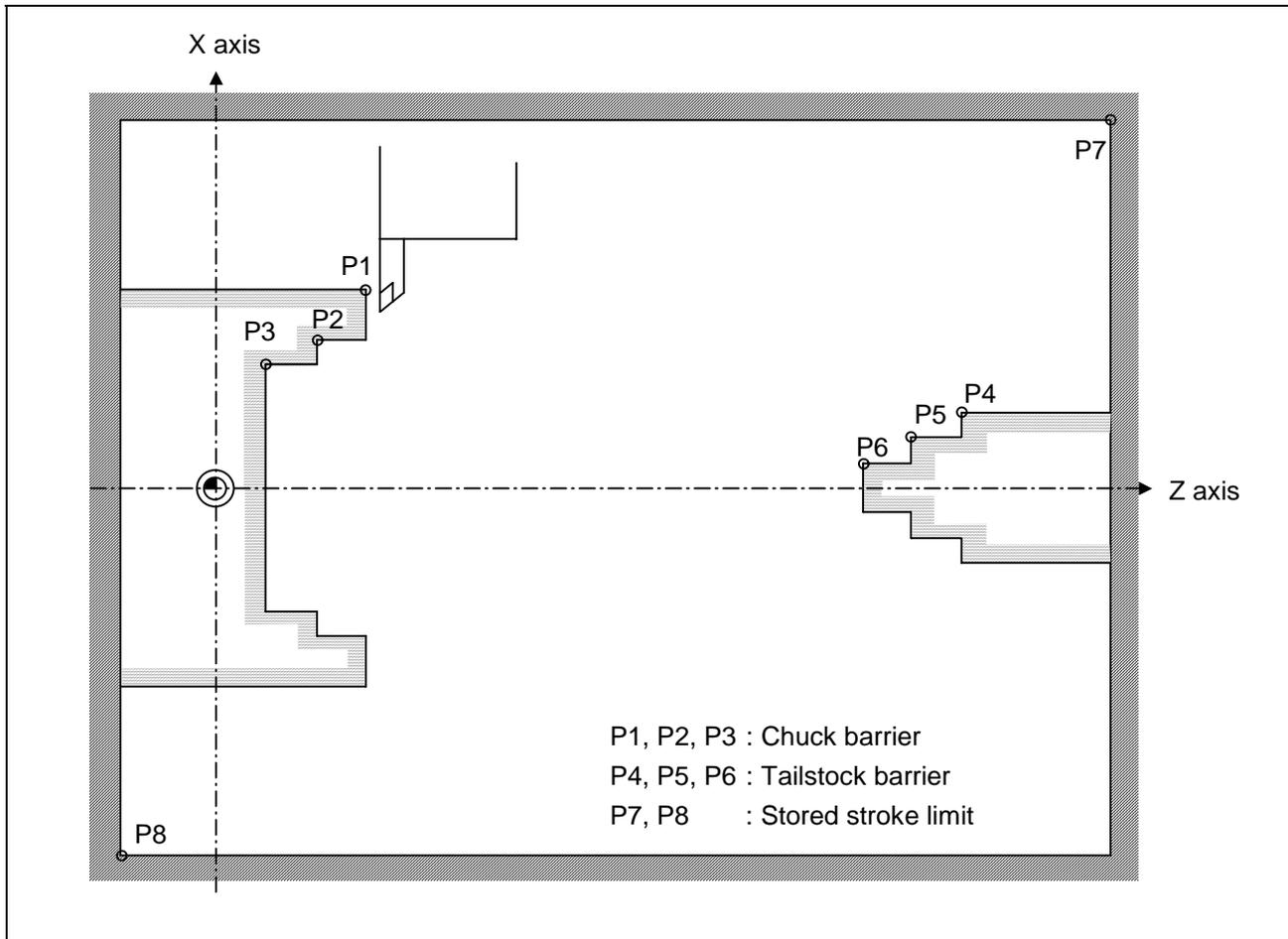
7.1.7 Precautions

- (1) If the maximum value and minimum value of the stored stroke limit's prohibited range are set to the same value, the following will occur.
 - When the maximum value and minimum value are set to 0, if the outside is the prohibited range, the entire range will be prohibited. If the inside is the prohibited range, the entire range will be the moveable range.
 - If data other than 0 is set for the maximum value and minimum value, the entire range will be the moveable range.
- (2) The stored stroke limit IC is valid when the axis parameter #2063 is changed.
If changed during automatic operation, the function will be validated after the smoothing for all axes reaches 0.
- (3) Make sure that the lower limit value of the stored stroke limit IC setting value is "smaller than the upper limit value".

7.2 Chuck Barrier/Tailstock Barrier (L System)

By limiting the tool nose movement range, the chuck barrier and tailstock barrier prevent collision with the chuck and tailstock due to programming errors. If movement is commanded which exceeds the region set in the parameters, the tool will automatically stop at the barrier boundary.

This function can be set by G code or PLC signal.

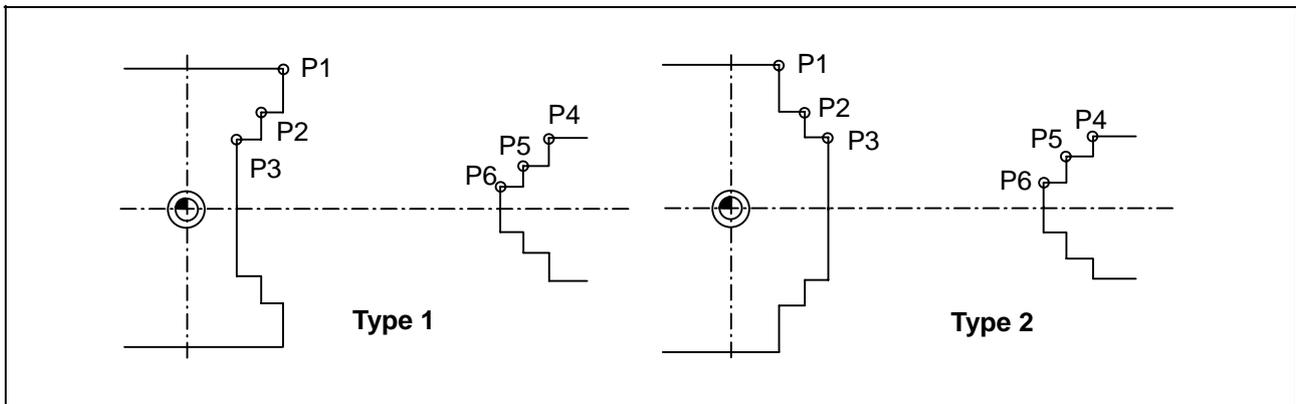


7.2.1 Detailed Description

- (1) An alarm will appear if the machine stops because it was about to exceed the set region.
Reset to cancel this alarm.
- (2) This function is also valid during machine lock.
- (3) This function is validated when all axes in which chuck barrier and tailstock barrier are set have finished their reference position returns.
- (4) When there is a stored stroke check function, and the stored stroke limit region is set, the chuck barrier/tailstock barrier function is validated simultaneously with the stored stroke check function.

7.2.2 Setting the Chuck Barrier/Tailstock Barrier

(1) Using G22 and G23



- (a) Three points can be input as parameters for both the chuck barrier and tailstock barrier. Set in the machine coordinate system.

Points P1, P2 and P3 (parameter "#8301 P1" to "#8303 P3") are for the chuck barrier. Points P4, P5 and P6 (parameter "#8304 P4" to "#8306 P6") are for the tailstock barrier.

- (b) The barrier region should be a symmetric shape regarding the Z axis. When the X axis coordinates of barrier point P_ are a negative value, reverse the sign to the positive side, then convert and check.

The absolute value of each barrier point's X axis coordinates must be set as follows.

$$P1 \geq P2 \geq P3, P4 \geq P5 \geq P6$$

(Note that the Z axis coordinates do not have to follow this setting.)

(2) Using the PLC external signal input

Set the coordinate values for each point between "#8300 P0" to "#8314 P10".

P0 is the basic X coordinate of the chuck and tailstock barrier. Set the workpiece center coordinate in the basic machine coordinate system as a radius value.

The coordinate value from the workpiece center (P0) is set as a radius value for the P1 to P10 X axes.

The Z axis is set as a basic machine coordinate system coordinate.

The barrier region is symmetrical in respect to P0.

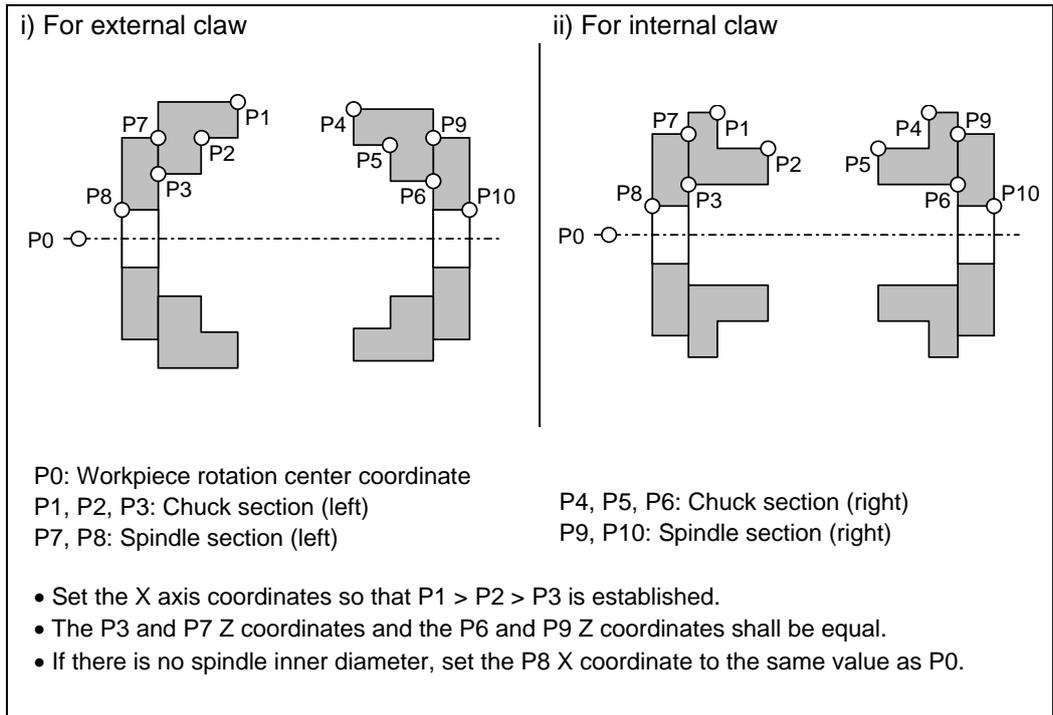
The X axis coordinate of each point must be set in the following manner.

$$\begin{array}{ll} P1 \geq P2 \geq P3 & P4 \geq P5 \geq P6 \\ P7 \geq P8 & P9 \geq P10 \end{array}$$

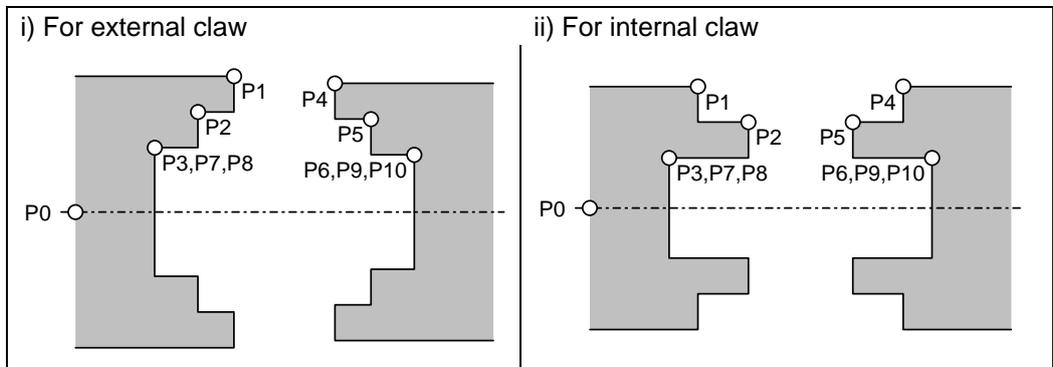
If the P8 Z axis coordinate must be less than P1 to P3, and the P10 Z axis coordinate must be more than P4 to P6.

The positions of the P0 to P10 points in the chuck/tailstock are as follow.

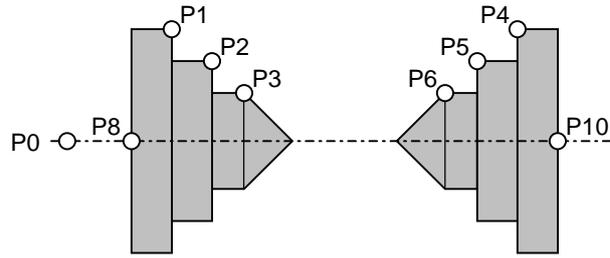
(a) Setting the chuck



If a spindle section is not to be set, set the P3, P7 and P8 values to the same values, and the P6, P9 and P10 values to the same values. The barrier range will be as shown below in this case.



(b) Setting the tailstock



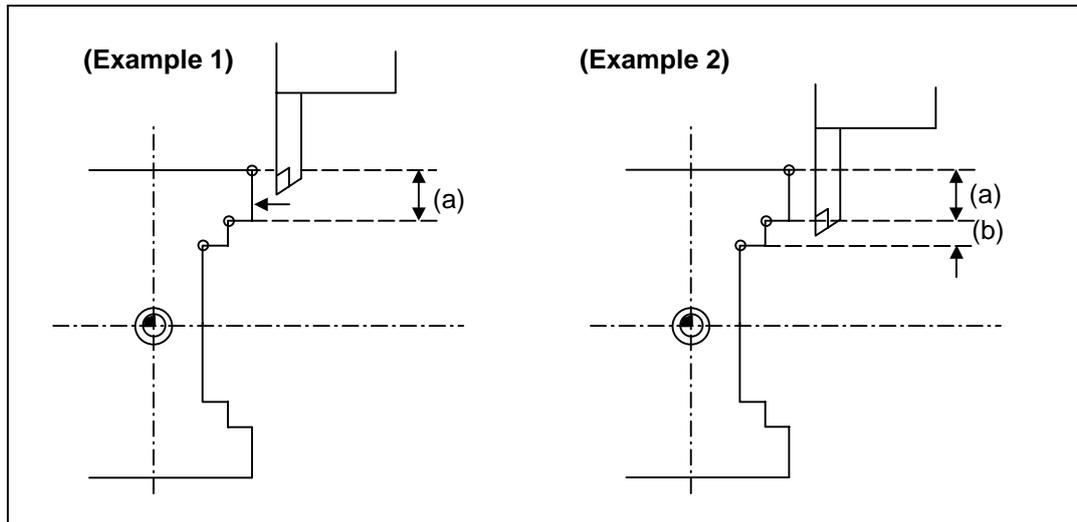
- P7 and P9 are not used.
- The tailstock end section angle is set with the parameters.
 - "#8318 STOCK ANGLE (L)"
 - "#8319 STOCK ANGLE (R)"
- If "0" is set, the angle will be 90°. (Default)
- Set the P8 X coordinate to the same value as P0. (No spindle inner diameter)

7.2.3 Restrictions

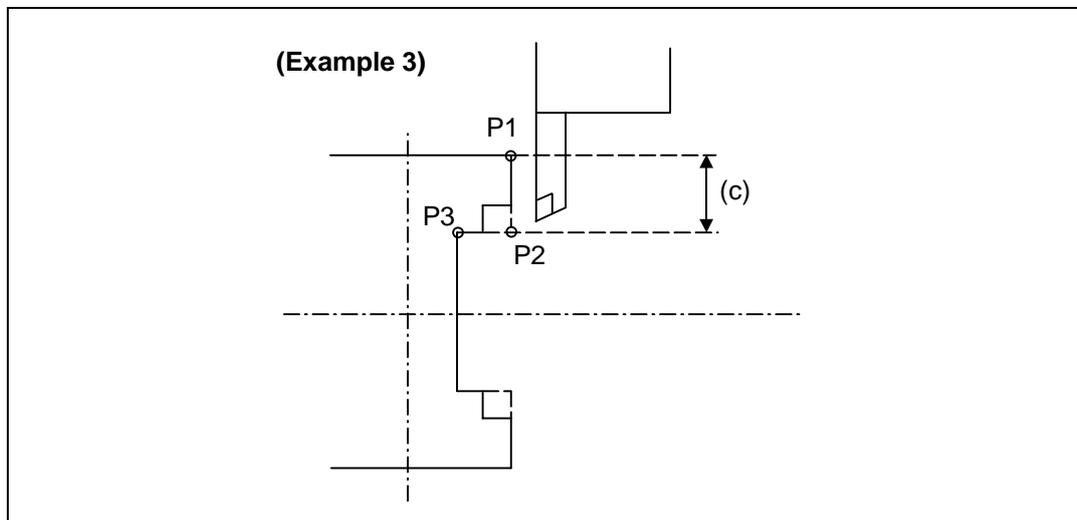
- (1) There is only one checkpoint from the tool regarding the chuck barrier/tailstock barrier. Therefore, the following cautions must be observed.

In the following examples, when the barrier points are set to be checked by the hypothetical tool nose point and the tool moves in the direction of the arrow in the drawing, the following situation may occur.

In Example 1, there is a checkpoint in the range "a", so the tool will automatically stop at the barrier boundary. However, in Example 2 there is a checkpoint in range "b", so the chuck and tool may collide in range "a".



To avoid this, Example 3 is given. In this example, if the barrier points P1, P2 and P3 are set and the checkpoint is set in range "c", the tool can be stopped at the barrier boundary.



- (2) When the tool enters the barrier region and an alarm occurs, the tool may move in the opposite direction from which it came, once the alarm is canceled by resetting.
- (3) There is no barrier region for axes without a reference position return function. Thus, there is no barrier alarm for that axis.
- (4) When the tool enters a canceled barrier region, and that barrier is then validated, an alarm will result immediately if the tool is moved.
If the alarm is canceled with reset in this manner, invalidate the barrier (G23) before escaping or change the value set for each barrier point.
- (5) The soft limit is valid even if the barrier is invalid (G23)

7.3 Computer Link B

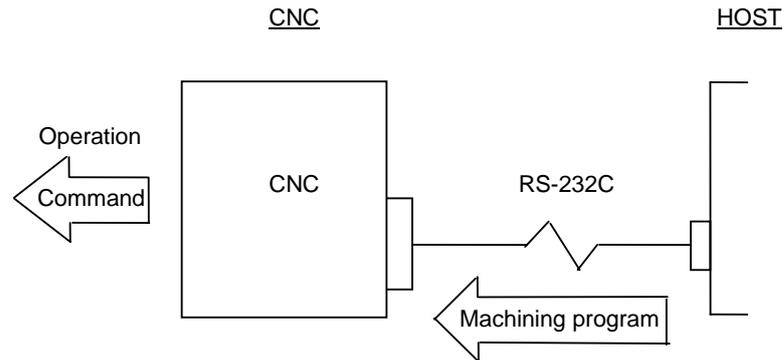
Computer link B is a function to receive/send data between the HOST computer (hereafter "HOST") and CNC.

This function sends transmission control character DC1 to the host computer at the CNC cycle start, and it enables operation to be performed while the machining programs are received from the HOST.

The computer link has a reception buffer so that operation will be less susceptible to the effects of the data transfer status at the HOST end.

The high-speed machining mode option is required for high-speed fine-segment machining.

This function cannot be operated in the 2nd and following part systems.

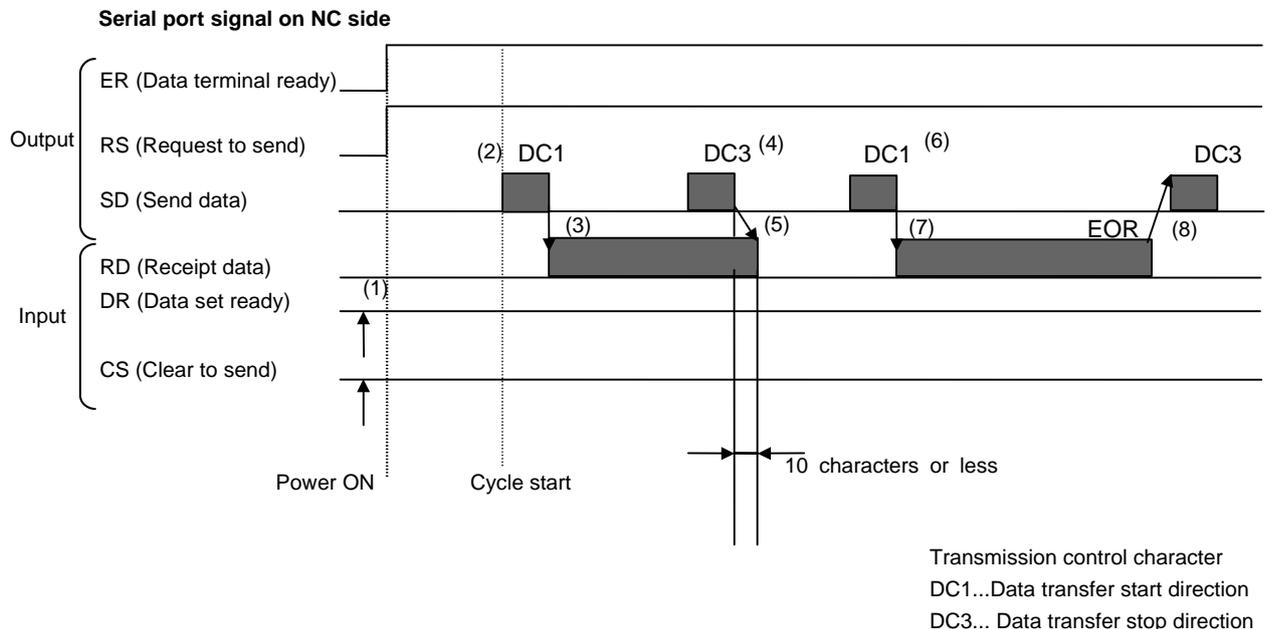


7.3.1 Communication Procedure

Communication method

The communication is carried out by the following flows.

- (1) Computer link B checks whether ER (DR of CNC) signal of HOST is turning ON.
(When the check result is OFF, the alarm "L01 Host ER signal OFF" appears.)
- (2) Computer link B sends **DC1** according to NC cycle start.
- (3) HOST starts the data transmission to computer link B after **DC1** was received.
- (4) When the receive buffer is full, computer link B sends **DC3** to HOST.
- (5) HOST stops the data transmission ten characters or less after **DC3** was received.
- (6) After the buffer process ends, Computer link B sends **DC1** to HOST again.
- (7) After receiving **DC1**, HOST sends the continuation of the previous data.
- (8) Computer link B executes the end processing by the data reading completion (received EOR).
(When the parameter "#9615 CTRL. CODE OUT/bit3(DC3 output)" is "1", **DC3** is sent to HOST when the communication ends.)

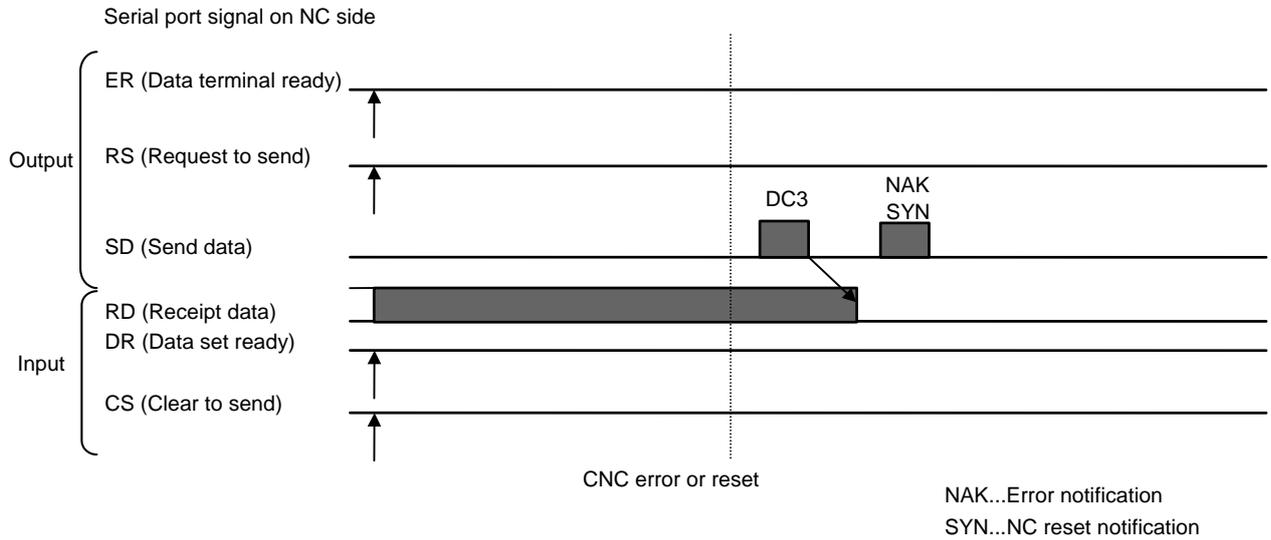


- Computer link B always carries out the request to send (RS) to HOST.
- After **DC1** is sent to HOST, computer link B ignores all receive data until the first EOB code is received, and buffering is executed from the data after EOB. The data reception is ended by EOR code reception.
- When the PC side serial port is used, the serial port opening of computer link B is executed at cycle start. When the operation ends, the port is closed.

Operation when error occurring and resetting

When the error occurs in CNC (program error and communication error) or when the reset is pressed, computer link B sends **DC3** to HOST.

When the parameter "#9615 CTRL. CODE OUT/bit0:NAK output)" is "1", **NAK** is sent to HOST after **DC3** is output.)



Parity V

When the parameter "#9610 LINK PARAM. 2/bit3 (Parity V)" is "1", checking of parity V is executed. As a result, if there is a character omission in the machining program from the host computer at the BTR operation, the program error and the computer link error are displayed and the operation stops.

7.4 Manual Synchronous Tapping

7.4.1 Outline

After a synchronous tap cycle is stopped with block stop or feed hold stop, this function carries out tapping by selecting and moving the drilling axis with manual handle operation.

7.4.2 Command Format

This function is commanded with the same format as for synchronous tapping.

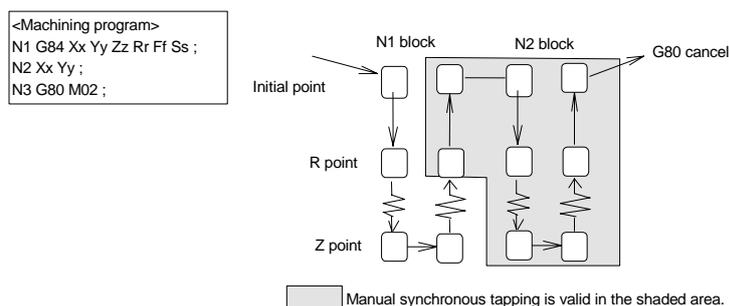
7.4.3 Operation Procedures

The procedures for manual synchronous tapping are described below.

- (1) When using the multiple spindle specifications, select the command spindle.
- (2) Stop the spindle and servo axis, and select the automatic operation mode.
- (3) Execute the synchronous tapping cycle program.
- (4) Apply block stop or feed hold stop.
- (5) Change to the handle mode. Select and move the drilling axis.
- (6) To cancel manual synchronous tapping, execute G80 in the same manner as the normal synchronous tap mode, or reset the program.

7.4.4 Precautions

- (1) When selecting the axis for the handle mode, select the same axis as that commanded in the program.
- (2) After executing the program with automatic operation, do not reset the program until manual synchronous tapping is completed.
The manual synchronous tapping mode will be canceled even if reset is applied in the handle mode.
- (3) During manual synchronous tapping, the acceleration/deceleration time constant is the same as normal handle operation. The return override is also invalid.
- (4) When using the multiple spindle specifications, changes to the commanded spindle selection will be validated after G80; is executed or reset is input to cancel synchronous tapping, and then manual synchronous tapping is started.
- (5) The spindle rotation speed during manual synchronous tapping is not clamped by the program's S command or parameter "#3013 stap1" to "#3016 stap4" (tap rotation speed).
- (6) Manual synchronous tapping is valid from the point that synchronous tapping and cutting is carried out once during automatic operation, and block stop is applied.



(Note) To validate manual synchronous tapping without carrying out cutting, run the following type of program with no movement commands.

```
G91 G84 X0 Y0 Z0 R0 F2. S1000 ;
```

III. MAINTENANCE

1. Daily Maintenance and Periodic Inspection and Maintenance

1.1 Maintenance Items

Maintenance is categorized into daily maintenance items (items to be carried at set intervals) and periodic maintenance items (replacement of parts when life is reached).

Some parts will not function in a hardware manner when the life is reached, so these should be replaced before the life is reached.

Class	Name	Life	Inspection/replacement	Remarks
Daily maintenance	Escutcheon		Once/two months (Accordingly when dirty)	
Periodic maintenance	Battery (lithium battery)	Cumulative data holding time: 45,000 hr	When battery voltage drop caution alarm occurs (Guideline: approx. 5 years)	Refer to "Control unit battery" in "1.2.1 Durable Parts"
	Cooling fan (700 Series only)	Control unit: 60,000 hr Display unit: 50,000 hr	Refer to left.	
	Back light	8.4-type: 700 Series: 30,000 hr 70 Series: 50,000 hr 10.4-type: 50,000 hr	Refer to left.	
	Hard disk unit (700 Series only)	20,000 hr or 5 years (Shorter one is applied.)	Refer to left.	
Other consumable parts	Operation board	10 ⁶ punches	Refer to left.	

1.1.1 Escutcheon

(1) Cleaning the escutcheon

- (a) Prepare the rear side of the escutcheon to clean.
- (b) Wipe the escutcheon with a soft, clean, dry cloth. If cleaning is still required, put some neutral detergent on a cloth and wipe. Do not use alcohol, thinner, etc.

1.1.2 LCD Panel

(1) Handling the LCD panel

(a) Precautions for use

- The polarizing plate (display surface) of the LCD panel surface can be easily scratched, so be careful during handling.
- Glass is used in the LCD panel. Be careful not to drop the LCD panel or allow it to hit hard objects, as the glass may chip or break.
- The polarizing plate may be stained or discolored if drops of water, etc., adhere to it for long periods, so be sure to wipe off any moisture immediately.
- Wipe off any dirt, dust, etc., on the polarizing plate using absorbent cotton or other soft cloth.
- A CMOS LSI is used in the LCD panel, so be careful of static electricity when handling.
- Never disassemble the LCD panel. Doing so will damage the panel.

(b) Precautions for storage

- Do not store the LCD panel in locations having a high temperature or humidity. (Store within the storage temperature range.)
- When storing the LCD panel as an individual unit, be sure that other objects do not touch or hit the polarizing plate (display surface).
- When storing the LCD panel for long periods, be sure to store in a dark place away from exposure to direct sunlight or fluorescent light.

(2) Other precautions for use

(a) Backlight life

The life of the backlight is as follows. (ambient temperature 25C°)

8.4-type : 30,000 hours (for 700 Series) or 50,000 hours (for 70 Series)

10.4-type : 50,000 hours

These are the time for luminance to drop to 50% of the initial value.

The backlight life is dependent on the temperature. The life tends to be shorter when used continuously at lower temperatures.

(b) Luminance start

Due to the characteristics of the backlight, the luminance could drop slightly at lower temperatures. It will take approx.10 to 15 minutes for the luminance to reach the rated value after the power is turned ON.

(c) Unevenness, luminescent spots and irregularities

Uneven brightness, small luminescent spots or small dark spots may appear on LCD panel, but this is not a fault.

(3) Replacing the backlight

Replacement should be commissioned to the Mitsubishi Service Center.

1.1.3 Compact Flash/IC card

(1) Handling the compact flash/IC card

The general handling methods for the compact flash/IC card are described below.

Refer to the instruction manual of the compact flash/IC card used for details.

(a) Precautions for use

- Insert the card in the correct direction.
- Do not touch the connector area with hands or metal.
- Do not apply excessive force to the connector area.
- Do not subject the card to bending or strong impacts.
- Do not open the cover or disassemble the card.
- Do not use the card in dusty locations.

(b) Precautions for storage

- Do not store the card in locations having a high temperature or humidity.
- Do not store the card in dusty locations.

1.2 Replacement Methods [700 series]

1.2.1 Durable Parts

Control unit battery

All data, such as the parameters and machining programs that need to be backed up when the power is turned OFF, are saved by a lithium battery installed in the control unit's battery holder.

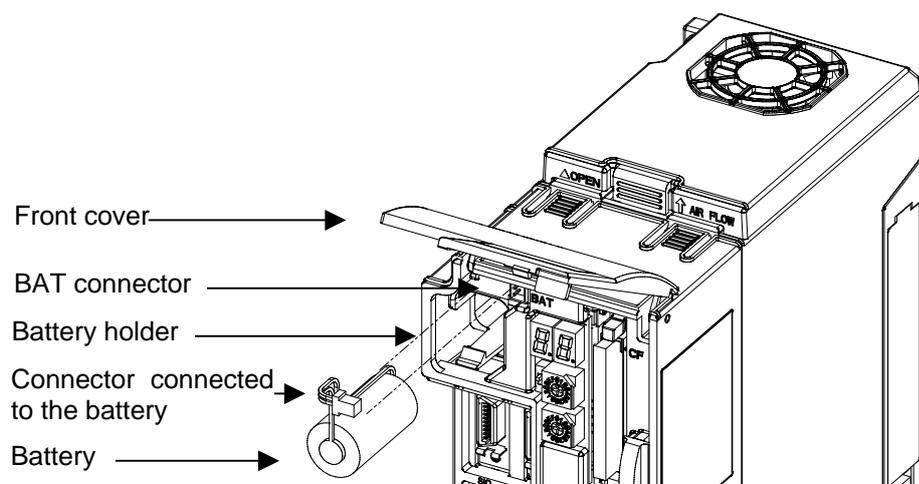
Battery	Q6BAT BKO-C10811H03 (SANYO CR17335SE-R with Mitsubishi specifications)
Battery cumulative data holding time	45,000 hours (At room temperature. The life will be shorter if the temperature is high.)
Battery life	Approx. 5 years (from date of battery manufacture)

(Note) Replace the battery when the alarm "Z52 Battery drop 0001" appears on the NC screen. The internal data could be damaged if the alarm "Z52 Battery drop 0003" appears.

[Replacement procedures]

Always replace the battery with the control unit (machine) power turned OFF.
Complete the replacement within 30 minutes after turning the power OFF. (If the battery is not connected within 30 minutes, the data being backed up may be destroyed.)

- (1) Check that the machine power is turned OFF. (If the power is not OFF, turn it OFF.)
- (2) Open the electric cabinet door, and confirm that the control unit LED, 7-segment display, etc., are all OFF.
- (3) Open the front cover of the control unit by pulling on the right side of the door.
- (4) Pull the connector connected to the battery out from the BAT connector.
- (5) Remove the battery from the battery holder.
- (6) Fit the new battery into the battery holder.
- (7) Insert the connector connected to the new battery into the BAT connector. Pay attention to the connector orientation, being careful not to insert backwards.
- (8) Close the front cover of the control unit. At this time, confirm that the cover is closed by listening for the "click" sound when the latch catches.
- (9) Close the door of the electric cabinet.



[Precautions for handling battery]

- (1) Always replace the battery with the same type.
- (2) Do not disassemble the battery.
- (3) Do not place the battery in flames or water.
- (4) Do not pressurize and deform the battery.
- (5) This is a primary battery so do not charge it.
- (6) Dispose of the spent battery as industrial waste.



CAUTION



If the battery low warning is issued, save the machining programs, tool data and parameters in an input/output device, and then replace the battery. When the battery alarm is issued, the machining programs, tool data and parameters may be destroyed. Reload the data after replacing the battery.



Do not short circuit, charge, overheat, incinerate or disassemble the battery.



Dispose the spent battery according to the local laws.

Cooling fan for control unit

Type109P0412H731

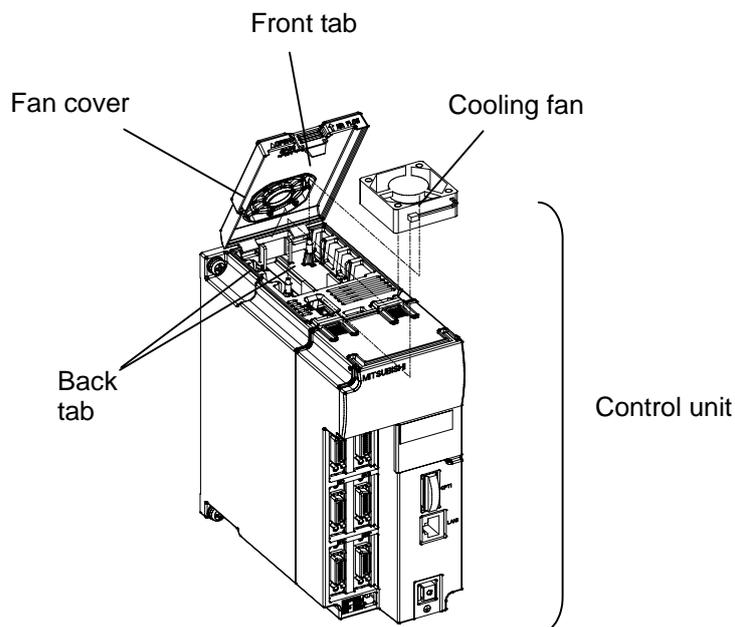
Life60,000 hours (When the rotary speed decreased 30% less than the initial values)

Life is a presumption value in 60°C environment, not guaranteed value.

[Replacement procedures]

Always replace the cooling fan with the control unit (machine) power turned OFF.

- (1) Check that the machine power is turned OFF. (If the power is not OFF, turn it OFF.)
- (2) Open the electric cabinet door.
- (3) Remove the fan cover on the top center of the control unit. (Remove the fan cover by pushing up the front tab.)
- (4) Pull out the cooling fan from the control unit cooling fan housing.
- (5) Pull the connector connected to the cooling fan out from the control unit PCB.
- (6) Replace the cooling fan with the new one. Insert the connector connected to the new cooling fan into the control unit PCB.
- (7) Put the cooling fan into the control section cooling fan housing. (Be sure the label side is on the top.)
- (8) Arrange the cooling fan wiring neatly in the control unit cooling fan housing.
- (9) Install the fan cover. (First install the back tabs in the control unit. Then install the front tabs in the control unit.)
- (10) Check the cables and wires for leaks of connection, faulty connections, etc., then close the electric cabinet door.



Caution

-  Do not replace the control unit while the power is ON.
-  Collect and dispose of the spent cooling fan according to the local laws.

Cooling fan for display unit (XP terminal)

TypeMMF-06D24DS-RP3

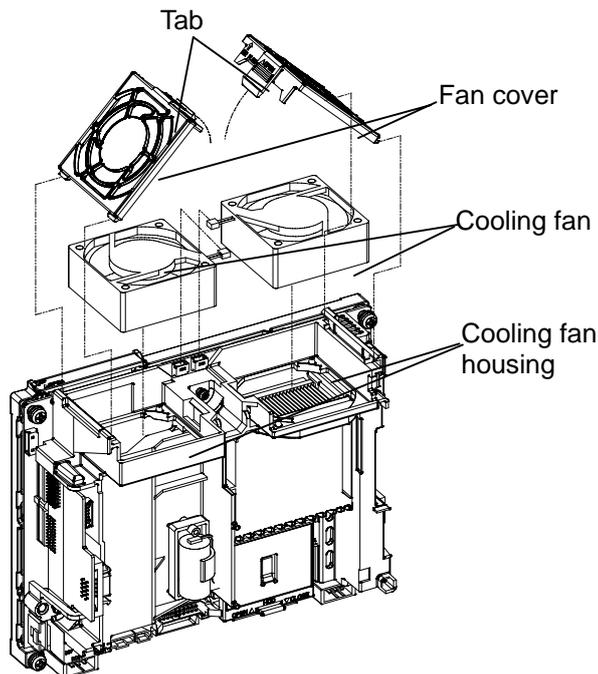
Life50,000 hours (When the rotary speed decreased 20% less than the initial values)

Life is a presumption value in 60°C environment, not guarantee value.

[Replacement procedures]

Always replace the cooling fan for display unit with the control unit (machine) power turned OFF.

- (1) Check that the machine power is turned OFF. (If the power is not OFF, turn it OFF.)
- (2) Open the electric cabinet door.
- (3) Remove the fan cover of cooling fan housing (2 parts) that is installed to PCB on back of display unit. (Remove the fan cover by pushing up the front tab.)
- (4) Pull out the cooling fan from the control unit cooling fan housing.
- (5) Pull the connector connected to the cooling fan out from the control unit PCB.
- (6) Replace the cooling fan with the new one. Insert the connector connected to the new cooling fan into the control unit PCB.
- (7) Put the cooling fan into the control section cooling fan housing. (Be sure the label side is on the top.)
- (8) Arrange the cooling fan wiring neatly in the control unit cooling fan housing.
- (9) Install the fan cover.
- (10) Check the cables and wires for leaks of connection, faulty connections, etc., then close the electric cabinet door.



Caution

-  Do not replace the control unit while the power is ON.
-  Collect and dispose of the spent cooling fan according to the local laws.

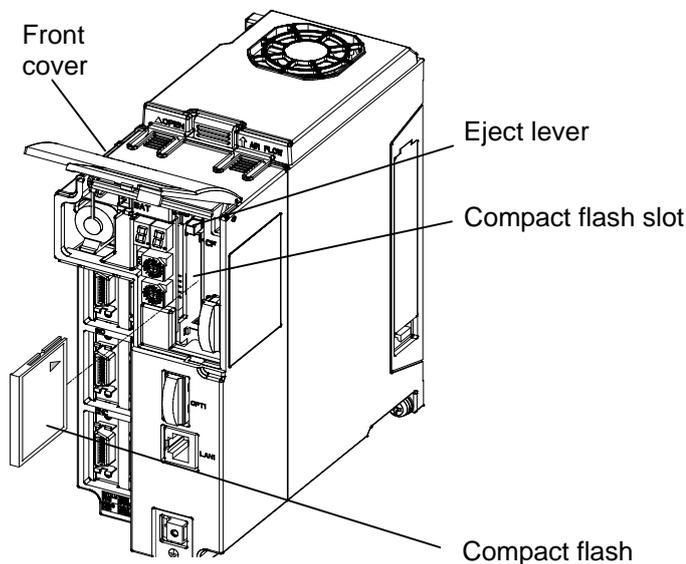
1.2.2 Compact Flash

Control unit compact flash

[Replacement procedures]

Always replace the compact flash with the control unit (machine) power turned OFF.

- (1) Check that the machine power is turned OFF. (If the power is not OFF, turn it OFF.)
- (2) Open the electric cabinet door, and confirm that the control unit LED, 7-segment display, etc., are all OFF.
- (3) Open the front cover of the control unit.
- (4) Press the eject lever twice to eject the compact flash.
- (5) Insert the new compact flash. (The surface is faced on the observers' right.)
- (6) Close the front cover of the control unit. At this time, confirm that the cover is closed by listening for the "click" sound when the latch catches.
- (7) Close the door of the electric cabinet.



(Note 1) There may be a compatibility problem between devices a commercially available compact flash memory, so illegal operations may occur.

1.2.3 IC Card

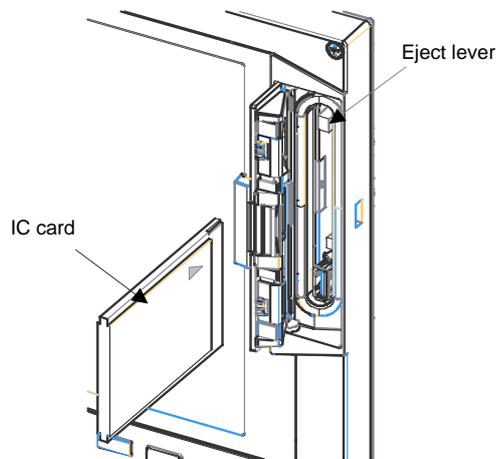
Front IC card

[Card insertion procedures]

- (1) Open the card slot door on display unit right end.
- (2) Insert the IC card. (The surface is faced on the observers' right.)

[Card replacement procedures]

- (1) Open the card slot door on display unit right end.
- (2) Press the eject lever twice to eject the IC card.



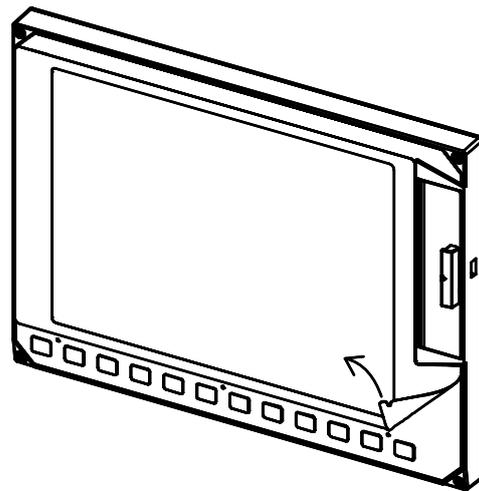
(Note 1) Do not eject an IC card during the data reading/writing.

(Note 2) There may be a compatibility problem between devices a commercially available IC card, so illegal operations may occur.

1.2.4 How to Replace the Protective Sheet on the Touch Panel

- (1) Remove the old protective sheet and clean the surface of the display unit.

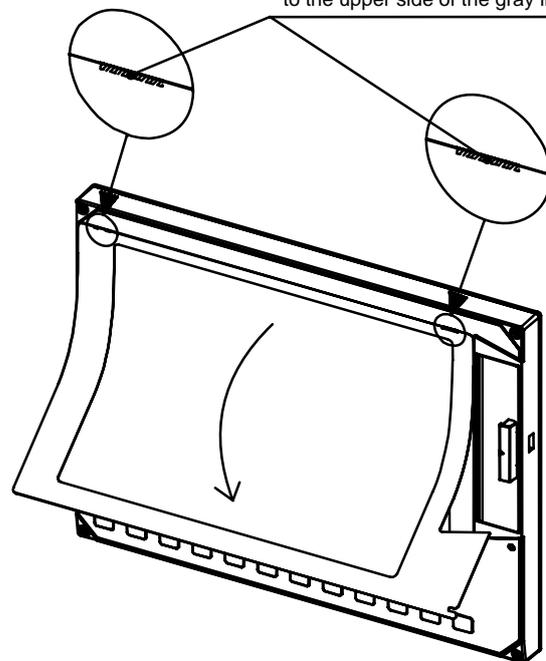
(Note) Peel off the protective sheet slowly.



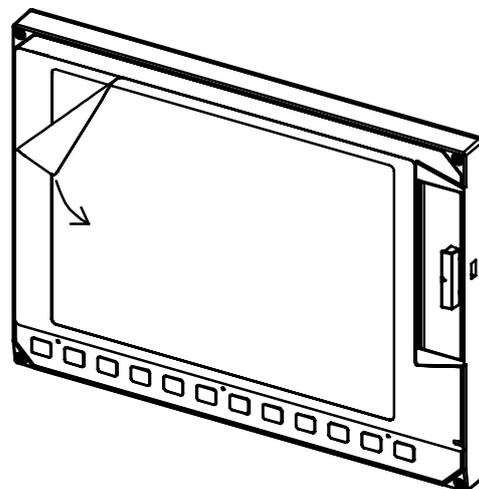
- (2) Remove the release paper on the back of the new protective sheet.

Fit the position of the sheet as shown in the figure and stick the sheet to the display unit.

Fit the edge of the protective sheet to the upper side of the gray line.



- (3) Remove the protective film on the surface of the protective sheet.



Touch panel for 10.4-type protective sheet type: BN939B036G51

1.3 Replacement Methods [70 series]

1.3.1 Durable Parts

Control unit battery

All data, such as the parameters and machining programs that need to be backed up when the power is turned OFF, are saved by a lithium battery installed in the control unit's battery holder.

Battery	Q6BAT BKO-C10811H03 (SANYO CR17335SE-R with Mitsubishi specifications)
Battery cumulative data holding time	45,000 hours (At room temperature. The life will be shorter if the temperature is high.)
Battery life	Approx. 5 years (from date of battery manufacture)

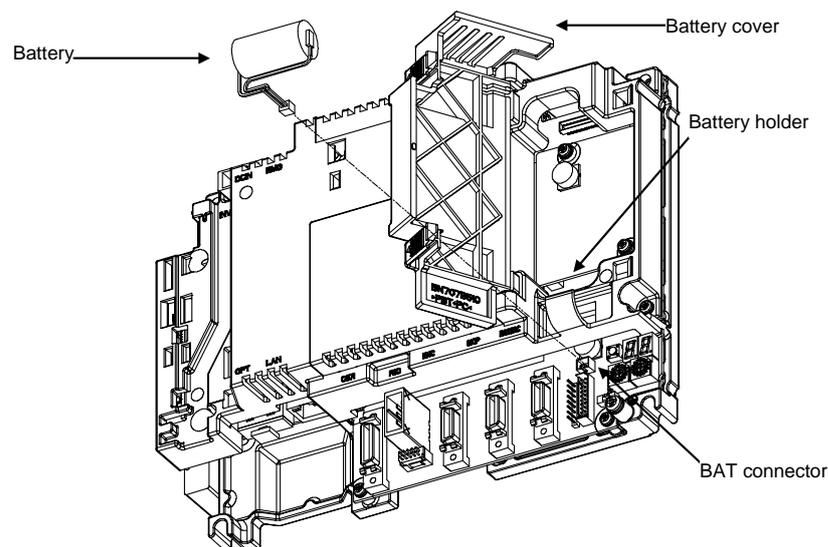
(Note) Replace the battery when the alarm "Z52 Battery drop 0001" appears on the NC screen.
The internal data could be damaged if the alarm "Z52 Battery drop 0003" appears.

[Replacement procedures]

Always replace the battery with the control unit (machine) power turned OFF.

Complete the replacement within 30 minutes after turning the power OFF. (If the battery is not connected within 30 minutes, the data being backed up might be destroyed.)

- (1) Check that the machine power is turned OFF. (If the power is not OFF, turn it OFF.)
- (2) Confirm that the control unit LED, 7-segment display, etc., are all OFF.
- (3) Open the battery cover of the control unit. Pull the right side of the battery cover toward front.
- (4) Pull the connector connected to the battery out from the BAT connector.
- (5) Remove the battery from the battery holder.
- (6) Fit the new battery into the battery holder.
- (7) Insert the connector connected to the new battery into the BAT connector. Pay attention to the connector orientation, being careful not to insert backwards.
- (8) Close the front cover of the control unit. At this time, confirm that the cover is closed by listening for the "click" sound when the latch catches.



[Precautions for handling battery]

- (1) Always replace the battery with the same type.
- (2) Do not disassemble the battery.
- (3) Do not place the battery in flames or water.
- (4) Do not pressurize and deform the battery.
- (5) This is a primary battery so do not charge it.
- (6) Dispose of the spent battery as industrial waste.



CAUTION

- ❗ If the battery low warning is issued, save the machining programs, tool data and parameters in an input/output device, and then replace the battery. When the battery alarm is issued, the machining programs, tool data and parameters may be destroyed. Reload the data after replacing the battery.
- ⚠ Do not short circuit, charge, overheat, incinerate or disassemble the battery.
- ⚠ Dispose the spent battery according to the local laws.

1.3.2 Compact Flash

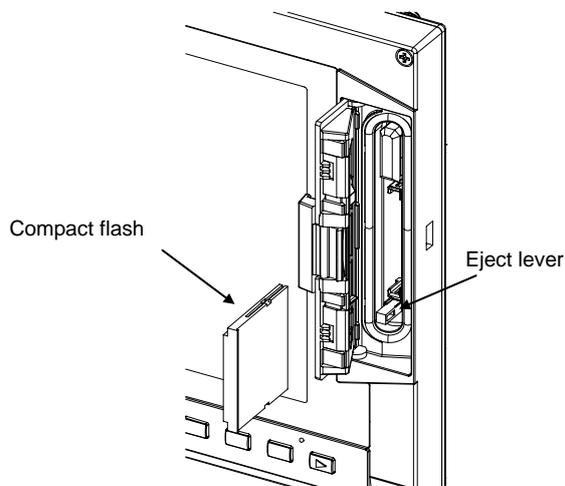
Front Compact Flash

[Card insertion procedures]

- (1) Open the card slot door located on the display unit right end.
- (2) Insert the compact flash. (The surface is faced on the observers' right.)

[Card ejecting procedures]

- (1) Open the card slot door located on the display unit right end.
- (2) Press the eject lever twice to eject the compact flash.



(Note 1) Do not eject a compact flash during the data reading/writing.

(Note 2) There may be a compatibility problem with non-recommended compact flashes, which may lead illegal operations.

1.3.3 How to Replace the Protective Sheet on the Touch Panel

Refer to "1.2.4 How to Replace the Protective Sheet on the Touch Panel".

IV. APPENDIXES

Appendix 1. List of Function Codes

Function code ISO	Control unit recognition	Subject to parity V count	Screen display	Setting and display unit key-in	Stored in memory	Punch-out output ISO	Internal NC system function
0 to 9	Yes	Counted	Displayed	Key-in	Stored	0 to 9	Numerical data
A to Z	Yes	Counted	Displayed	Key-in	Stored	A to Z	Addresses
+	Yes	Counted	Displayed	Key-in	Stored	+	Sign, variable operator (+)
-	Yes	Counted	Displayed	Key-in	Stored	-	Sign, variable operator (-)
.	Yes	Counted	Displayed	Key-in	Stored	.	Decimal point
,	Yes	Counted	Displayed	Key-in	Stored	,	
/	Yes	Counted	Displayed	Key-in	Stored	/	Block delete (optional block skip) Variable operator (÷)
%	Yes	Counted	Displayed (%)	No key-in (automatically inserted)	Stored	%	End of record (tape storage end) Rewind start & stop during tape search
LF/NL	Yes	Counted	Displayed (:)	Key-in ;/EOB	Stored	LF	End of block
(Yes	Counted	Displayed	Key-in	Stored	(Control out (comment start)
)	Yes	Counted	Displayed	Key-in	Stored)	Control in (comment end)
:	Yes	Counted	Displayed	No key-in	Stored	:	Program number address (instead of O.)
#	Yes	Counted	Displayed	Key-in	Stored	#	Variable number
*	Yes	Counted	Displayed	Key-in	Stored	*	Variable operator (X)
=	Yes	Counted	Displayed	Key-in	Stored	=	Variable definition
[Yes	Counted	Displayed	Key-in	Stored	[Variable operator
]	Yes	Counted	Displayed	Key-in	Stored]	Variable operator
BS	No		Not displayed	No key-in	Stored		
HT	No		Not displayed	No key-in	Stored		
SP	No		Not displayed	Key-in	Stored	SP (T-V automatic adjustment)	
CR	No		Not displayed	No key-in	Stored		
DEL	No	Not counted	Not displayed	No key-in	Not stored		
NULL	No	Not counted	Not displayed	No key-in	Not stored		
Any other	No	Counted	(Note 2)	No key-in	Stored		

(Note 1) Codes not listed in the above table are stored on tape, but an error will result during operation if they are not comments.

(Note 2) This denotes characters (including blanks) which are stored inside and which correspond to the command codes. "@" is not displayed.

Appendix 2. Table of Command Value Ranges

<Table of command values and setting value ranges>

(1) Linear axis: Input increment [mm] (M system)

Least command increment	0.001	0.0001	0.00001	0.000001
Maximum stroke (value for machine coordinate system)	±99999.999 mm	±99999.9999 mm	±99999.99999 mm	±99999.999999 mm
Maximum programmable dimension	±99999.999 mm	±99999.9999 mm	±99999.99999 mm	±99999.999999 mm
Rapid traverse rate (including dry run)	1 to 1000000 mm/min	1 to 1000000 mm/min	1 to 1000000 mm/min	1 to 1000000 mm/min
Cutting feed rate (including dry run) Asynchronous feed (feed per minute)	0.001 to 1000000.000 mm/min	0.0001 to 1000000.0000 mm/min	0.00001 to 1000000.00000 mm/min	0.000001 to 1000000.000000 mm/min
Synchronous feed (feed per revolution)	0.001 to 999.999 mm/rev	0.0001 to 999.9999 mm/rev	0.00001 to 999.99999 mm/rev	0.000001 to 999.999999 mm/rev
2nd to 4th reference position compensation (value for machine coordinate system)	±99999.999 mm	±99999.9999 mm	±99999.99999 mm	±99999.999999 mm
Tool compensation amount (shape)	±99999.999 mm	±99999.9999 mm	±99999.99999 mm	±99999.999999 mm
Tool compensation amount (wear)	±99999.999 mm	±99999.9999 mm	±99999.99999 mm	±99999.999999 mm
Incremental feed amount	0.001 mm/pulse	0.0001 mm/pulse	0.00001 mm/pulse	0.000001 mm/pulse
Handle feed amount	0.001 mm/pulse	0.0001 mm/pulse	0.00001 mm/pulse	0.000001 mm/pulse
Soft limit range (value with machine coordinate system)	±99999.999 mm	±99999.9999 mm	±99999.99999 mm	±99999.999999 mm
Dwell time	0 to 99999.999 s	0 to 99999.9999 s	0 to 99999.99999 s	0 to 99999.999999 s
Backlash compensation amount	±9999999 pulse	±9999999 pulse	±9999999 pulse	±9999999 pulse
Pitch error compensation amount	-32768 to 32767 pulse	-32768 to 32767 pulse	-32768 to 32767 pulse	-32768 to 32767 pulse
Thread lead (F)	0.001 to 999.999 mm/rev	0.0001 to 999.9999 mm/rev	0.00001 to 999.99999 mm/rev	0.000001 to 999.999999 mm/rev
Thread lead (precision E)	0.00001 to 999.99999 mm/rev	0.000001 to 999.999999 mm/rev	0.0000001 to 999.9999999 mm/rev	0.00000001 to 999.99999999 mm/rev
Thread lead (ridges/inch)	0.03 to 999.99	0.026 to 999.999	0.0255 to 999.9999	0.02541 to 999.99999

Appendix 2. Table of Command Value Ranges

(2) Linear axis: Input increment [inch] (M system)

Least command increment	0.0001	0.00001	0.000001	0.0000001
Maximum stroke (value for machine coordinate system)	±9999.9999 inch	±9999.99999 inch	±9999.999999 inch	±9999.9999999 inch
Maximum programmable dimension	±9999.9999 inch	±9999.99999 inch	±9999.999999 inch	±9999.9999999 inch
Rapid traverse rate (including dry run)	1 to 100000 inch/min	1 to 100000 inch/min	1 to 100000 inch/min	1 to 100000 inch/min
Cutting feed rate (including dry run) Asynchronous feed (feed per minute)	0.0001 to 100000.0000 inch/min	0.00001 to 100000.00000 inch/min	0.000001 to 100000.000000 inch/min	0.0000001 to 100000.0000000 inch/min
Synchronous feed (feed per revolution)	0.0001 to 999.9999 inch/rev	0.00001 to 999.99999 inch/rev	0.000001 to 999.999999 inch/rev	0.0000001 to 999.9999999 inch/rev
2nd to 4th reference position compensation (value for machine coordinate system)	±9999.9999 inch	±9999.99999 inch	±9999.999999 inch	±9999.9999999 inch
Tool compensation amount (shape)	±9999.9999 inch	±9999.99999 inch	±9999.999999 inch	±9999.9999999 inch
Tool compensation amount (wear)	±9999.9999 inch	±9999.99999 inch	±9999.999999 inch	±9999.9999999 inch
Incremental feed amount	0.0001 inch/pulse	0.00001 inch/pulse	0.000001 inch/pulse	0.0000001 inch/pulse
Handle feed amount	0.0001 inch/pulse	0.00001 inch/pulse	0.000001 inch/pulse	0.0000001 inch/pulse
Soft limit range (value with machine coordinate system)	±9999.9999 inch	±9999.99999 inch	±9999.999999 inch	±9999.9999999 inch
Dwell time	0 to 99999.999 s	0 to 99999.9999 s	0 to 99999.99999 s	0 to 99999.999999 s
Backlash compensation amount	±9999999 pulse	±9999999 pulse	±9999999 pulse	±9999999 pulse
Pitch error compensation amount	-32768 to 32767 pulse	-32768 to 32767 pulse	-32768 to 32767 pulse	-32768 to 32767 pulse
Thread lead (F)	0.0001 to 99.9999 inch/rev	0.00001 to 99.99999 inch/rev	0.000001 to 99.999999 inch/rev	0.0000001 to 99.9999999 inch/rev
Thread lead (precision E)	0.000001 to 39.370078 inch/rev	0.0000001 to 39.3700787 inch/rev	0.00000001 to 39.37007874 inch/rev	0.000000001 to 39.370078740 inch/rev
Thread lead (ridges/inch)	0.0101 to 9999.9999	0.01001 to 9999.99999	0.010001 to 9999.999999	0.0100001 to 9999.9999999

Appendix 2. Table of Command Value Ranges

(3) Rotation axis: degree [°] (M system)

Least command increment	0.001	0.0001	0.00001	0.000001
Maximum stroke (value for machine coordinate system)	±99999.999 °	±99999.9999 °	±99999.99999 °	±99999.999999 °
Maximum programmable dimension	±99999.999 °	±99999.9999 °	±99999.99999 °	±99999.999999 °
Rapid traverse rate (including dry run)	1 to 1000000 °/min	1 to 1000000 °/min	1 to 1000000 °/min	1 to 1000000 °/min
Cutting feed rate (including dry run) Asynchronous feed (feed per minute)	0.001 to 1000000.000 °/min	0.0001 to 1000000.0000 °/min	0.00001 to 1000000.00000 °/min	0.000001 to 1000000.000000 °/min
Synchronous feed (feed per revolution)	0.001 to 999.999 °/rev	0.0001 to 999.9999 °/rev	0.00001 to 999.99999 °/rev	0.000001 to 999.999999 °/rev
2nd to 4th reference position compensation (value with machine coordinate system)	±99999.999 °	±99999.9999 °	±99999.99999 °	±99999.999999 °
Incremental feed amount	0.001 °/pulse	0.0001 °/pulse	0.00001 °/pulse	0.000001 °/pulse
Handle feed amount	0.001 °/pulse	0.0001 °/pulse	0.00001 °/pulse	0.000001 °/pulse
Soft limit range (value with machine coordinate system)	±99999.999 °	±99999.9999 °	±99999.99999 °	±99999.999999 °
Backlash compensation amount	±9999999 pulse	±9999999 pulse	±9999999 pulse	±9999999 pulse
Pitch error compensation amount	-32768 to 32767 pulse	-32768 to 32767 pulse	-32768 to 32767 pulse	-32768 to 32767 pulse

Appendix 2. Table of Command Value Ranges

<Table of command values and setting value ranges>

(1) Linear axis: Input increment [mm] (L system)

Least command increment	0.001	0.0001	0.00001	0.000001
Maximum stroke (value for machine coordinate system)	±99999.999 mm	±99999.9999 mm	±99999.99999 mm	±99999.999999 mm
Maximum programmable dimension	±99999.999 mm	±99999.9999 mm	±99999.99999 mm	±99999.999999 mm
Rapid traverse rate (including dry run)	1 to 1000000 mm/min	1 to 1000000 mm/min	1 to 1000000 mm/min	1 to 1000000 mm/min
Cutting feed rate (including dry run) Asynchronous feed (feed per minute)	0.001 to 1000000.000 mm/min	0.0001 to 1000000.0000 mm/min	0.00001 to 1000000.00000 mm/min	0.000001 to 1000000.000000 mm/min
Synchronous feed (feed per revolution)	0.0001 to 999.9999 mm/rev	0.00001 to 999.99999 mm/rev	0.000001 to 999.999999 mm/rev	0.0000001 to 999.9999999 mm/rev
2nd to 4th reference position compensation (value for machine coordinate system)	±99999.999 mm	±99999.9999 mm	±99999.99999 mm	±99999.999999 mm
Tool compensation amount (shape)	±99999.999 mm	±99999.9999 mm	±99999.99999 mm	±99999.999999 mm
Tool compensation amount (wear)	±99999.999 mm	±99999.9999 mm	±99999.99999 mm	±99999.999999 mm
Incremental feed amount	0.001 mm/pulse	0.0001 mm/pulse	0.00001 mm/pulse	0.000001 mm/pulse
Handle feed amount	0.001 mm/pulse	0.0001 mm/pulse	0.00001 mm/pulse	0.000001 mm/pulse
Soft limit range (value with machine coordinate system)	±99999.999 mm	±99999.9999 mm	±99999.99999 mm	±99999.999999 mm
Dwell time	0 to 99999.999 s	0 to 99999.9999 s	0 to 99999.99999 s	0 to 99999.999999 s
Backlash compensation amount	±9999999 pulse	±9999999 pulse	±9999999 pulse	±9999999 pulse
Pitch error compensation amount	-32768 to 32767 pulse	-32768 to 32767 pulse	-32768 to 32767 pulse	-32768 to 32767 pulse
Thread lead (F)	0.001 to 999.999 mm/rev	0.0001 to 999.9999 mm/rev	0.00001 to 999.99999 mm/rev	0.000001 to 999.999999 mm/rev
Thread lead (precision E)	0.00001 to 999.99999 mm/rev	0.000001 to 999.999999 mm/rev	0.0000001 to 999.9999999 mm/rev	0.00000001 to 999.99999999 mm/rev
Thread lead (ridges/inch)	0.03 to 9999.99	0.026 to 9999.999	0.0255 to 9999.9999	0.02541 to 9999.99999

Appendix 2. Table of Command Value Ranges

(2) Linear axis: Input increment [inch] (L system)

Least command increment	0.0001	0.00001	0.000001	0.0000001
Maximum stroke (value for machine coordinate system)	±9999.9999 inch	±9999.99999 inch	±9999.999999 inch	±9999.9999999 inch
Maximum programmable dimension	±9999.9999 inch	±9999.99999 inch	±9999.999999 inch	±9999.9999999 inch
Rapid traverse rate (including dry run)	1 to 100000 inch/min	1 to 100000 inch/min	1 to 100000 inch/min	1 to 100000 inch/min
Cutting feed rate (including dry run) Asynchronous feed (feed per minute)	0.0001 to 100000.0000 inch/min	0.00001 to 100000.00000 inch/min	0.000001 to 100000.000000 inch/min	0.0000001 to 100000.0000000 inch/min
Synchronous feed (feed per revolution)	0.000001 to 99.999999 inch/rev	0.0000001 to 99.9999999 inch/rev	0.00000001 to 99.99999999 inch/rev	0.000000001 to 99.999999999 inch/rev
2nd to 4th reference position compensation (value for machine coordinate system)	±9999.9999 inch	±9999.99999 inch	±9999.999999 inch	±9999.9999999 inch
Tool compensation amount (shape)	±9999.9999 inch	±9999.99999 inch	±9999.999999 inch	±9999.9999999 inch
Tool compensation amount (wear)	±9999.9999 inch	±9999.99999 inch	±9999.999999 inch	±9999.9999999 inch
Incremental feed amount	0.0001 inch/pulse	0.00001 inch/pulse	0.000001 inch/pulse	0.0000001 inch/pulse
Handle feed amount	0.0001 inch/pulse	0.00001 inch/pulse	0.000001 inch/pulse	0.0000001 inch/pulse
Soft limit range (value with machine coordinate system)	±9999.9999 inch	±9999.99999 inch	±9999.999999 inch	±9999.9999999 inch
Dwell time	0 to 99999.999 s	0 to 99999.9999 s	0 to 99999.99999 s	0 to 99999.999999 s
Backlash compensation amount	±9999999 pulse	±9999999 pulse	±9999999 pulse	±9999999 pulse
Pitch error compensation amount	-32768 to 32767 pulse	-32768 to 32767 pulse	-32768 to 32767 pulse	-32768 to 32767 pulse
Thread lead (F)	0.0001 to 99.9999 inch/rev	0.00001 to 99.99999 inch/rev	0.000001 to 99.999999 inch/rev	0.0000001 to 99.9999999 inch/rev
Thread lead (precision E)	0.000001 to 39.370078 inch/rev	0.0000001 to 39.3700787 inch/rev	0.00000001 to 39.37007874 inch/rev	0.000000001 to 39.370078740 inch/rev
Thread lead (ridges/inch)	0.0101 to 9999.9999	0.01001 to 9999.99999	0.010001 to 9999.999999	0.0100001 to 9999.9999999

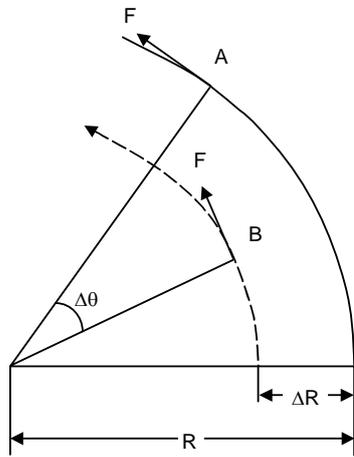
Appendix 2. Table of Command Value Ranges

(3) Rotation axis: degree [°] (L system)

Least command increment	0.001	0.0001	0.00001	0.000001
Maximum stroke (value for machine coordinate system)	±99999.999 °	±99999.9999 °	±99999.99999 °	±99999.999999 °
Maximum programmable dimension	±99999.999 °	±99999.9999 °	±99999.99999 °	±99999.999999 °
Rapid traverse rate (including dry run)	1 to 1000000 °/min	1 to 1000000 °/min	1 to 1000000 °/min	1 to 1000000 °/min
Cutting feed rate (including dry run) Asynchronous feed (feed per minute)	0.001 to 1000000.000 °/min	0.0001 to 1000000.0000 °/min	0.00001 to 1000000.00000 °/min	0.000001 to 1000000.000000 °/min
Synchronous feed (feed per revolution)	0.0001 to 999.9999 °/rev	0.00001 to 999.99999 °/rev	0.000001 to 999.999999 °/rev	0.0000001 to 999.9999999 °/rev
2nd to 4th reference position compensation (value with machine coordinate system)	±99999.999 °	±99999.9999 °	±99999.99999 °	±99999.999999 °
Incremental feed amount	0.001 °/pulse	0.0001 °/pulse	0.00001 °/pulse	0.000001 °/pulse
Handle feed amount	0.001 °/pulse	0.0001 °/pulse	0.00001 °/pulse	0.000001 °/pulse
Soft limit range (value with machine coordinate system)	±99999.999 °	±99999.9999 °	±99999.99999 °	±99999.999999 °
Backlash compensation amount	±9999999 pulse	±9999999 pulse	±9999999 pulse	±9999999 pulse
Pitch error compensation amount	-32768 to 32767 pulse	-32768 to 32767 pulse	-32768 to 32767 pulse	-32768 to 32767 pulse

Appendix 3. Circular Cutting Radius Error

When circular cutting is performed, an error is caused between the command coordinate and the tracking coordinate due to the tracking delay in the smoothing circuit and servo system, and the workpiece ends up with a radius smaller than the commanded value. The method for obtaining this error (radius error) is shown below.



- A : Command coordinate
- B : Tracking coordinate
- R : Command radius (mm)
- ΔR : Radius error (mm)
- $\Delta\theta$: Angle error (rad)
- F : Cutting feed rate (m/min)

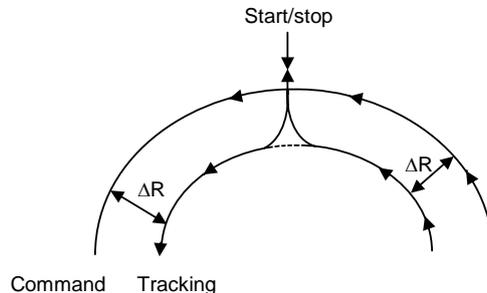
The radius error ΔR and angle error $\Delta\theta$ are calculated from the following formula.

Exponential acceleration/ deceleration	$\Delta R = \frac{1}{R} \cdot \left(\frac{1}{2} T_s^2 + \frac{1}{2} T_p^2 \right) \cdot \left(\frac{F \times 10^3}{60} \right)^2$ (mm)
Linear acceleration/ deceleration	$\Delta R = \frac{1}{R} \cdot \left(\frac{1}{24} T_s^2 + \frac{1}{2} T_p^2 \right) \cdot \left(\frac{F \times 10^3}{60} \right)^2$ (mm)

$$\Delta\theta = \tan^{-1} \left(T_s \cdot \frac{F}{R} \right) + \tan^{-1} \left(T_p \cdot \frac{F}{R} \right) \text{ (rad)}$$

T_s : Time constant (s) of specified smoothing circuit
 T_p : Position loop time constant

- (Note 1)** When the ΔR radius error applying with circular cutting does not come within the allowable value, proceed to reduce the cutting feed rate F, set T_s to a lower value or review the program.
- (Note 2)** In the steady state, ΔR is constant. However, it is not constant with command start and stop transitions. Under command start and stop conditions, therefore, the tracking coordinate should be as shown in the figure below.



Appendix 4. Registering/Editing the Fixed Cycle Program

The subprogram for the fixed cycle can be input, output and edited.



CAUTION



Do not change the fixed cycle program without prior approval from the machine maker.

4.1 Fixed Cycle Operation Parameters

To input/output or edit the data of each fixed cycle subprogram, use the Data I/O and Edit screens in the same way as when creating usual user-created work programs. In this case, the parameters must have been set. Set "1" in parameter "#1166 fixpro". If this parameter is valid, the IN/OUT and Edit screens are usable only for operating a fixed cycle control subprogram. During this period, program file displays only fixed cycle programs. Thus, after fixed cycle program operation, return parameter to "0".

(Note) Parameter "#1166 fixpro" will be set to 0 when the power is turned OFF.

4.2 Transmitting/Erasing the Fixed Cycle Program

Transmit/erase the fixed cycle program from the Data I/O screen. Check that fixed cycle operation parameter "#1166 fixpro" is valid.

The operating procedure is the same as a user machining program.

4.3 Standard Fixed Cycle Subprogram (For L system)

G37 (O370)	Automatic tool length measurement
------------	-----------------------------------

```
G31 Z #5 F #3 ;
1F [ROUND [ ABS [#2 - [ ##10 * #11 - #12 ] ] ] GT #8 ] GOTO 1 ;
1F [ROUND [ ##10 * #11 - #12 ] EQ #4 ] GOTO 1 ;
##9 = #10 - #12/#11 - #2/#11 + ##9 ;
#3003 = #1 ;
N2 ;
M99 ;
N1 # 3901 = 126 ;
```

G74 (O740)	End face cutoff cycle
------------	-----------------------

```
G 1 ;
1F [ ABS [ #2 ] GT 0 ] GOTO 10 ;
#14 = 1 ;
N10 #13 = #3 ;
IF [ #15 NE 0 ] GOTO 11 ;
#13 = #3 - #5 ;
N11 #16 = 0 ;
D0 1 ;
#10 = 0 ;
#11 = #4 ;
D0 2 ;
#10 = #10 + #4 ;
IF [ ABS [ #10 ] GE [ ABS [ #1 ] ] ] GOTO 1 ;
G01 X #11 ;
G00 X #6 ;
#11 = #4 - #6 ;
END 2 ;
N1 G01 X#1 - #10 + #11 ;
IF [ #15 EQ 0 ] GOTO 20 ;
IF [ #16 EQ 0 ] GOTO 21 ;
N20 G00 Y#5 ;
N21 #16 = 1 ;
G00X - #1 ;
IF [ #14 ] GOTO 3 ;
#12 = #12 + #3 ;
IF [ ABS [ #12 ] LT [ ABS [ #2 ] ] ] GOTO 2 ;
#14 = 1 ;
#13 = #2 - #12 + #13 ;
N2 G00 Y #13 ;
#13 = #3 - #5 ;
END 1 ;
N3 G00 Y - #2 - #5 ;
M99 ;
```

Appendix 4. Registering/Editing the Fixed Cycle Program

4.3 Standard Fixed Cycle Subprogram (For L system)

G75 (O750)	Longitudinal cutting cycle
------------	----------------------------

```
G. 1 ;
1F [ ABS [ #1 ] GT 0 ] GOTO 10 ;
#14 = 1 ;
N10 #13 = #4 ;
IF [ #15 NE 0 ] GOTO 11 ;
#13 = #4 - #5 ;
N11 #16 = 0 ;
D0 1 ;
#10 = 0 ;
#11 = #3 ;
D0 2 ;
#10 = #10 + #3 ;
IF [ ABS [ #10 ] GE [ ABS [ #2 ] ] ] GOTO 1 ;
G01 Y #11 ;
G00 Y #6 ;
#11 = #3 - #6 ;
END 2 ;
N1 G01 Y#2 - #10 + #11 ;
IF [ #15 EQ 0 ] GOTO 20 ;
IF [ #16 EQ 0 ] GOTO 21 ;
N20 G00 X#5 ;
N21 #16 = 1 ;
G00Y - #2 ;
IF [ #14 ] GOTO 3 ;
#12 = #12 + #4 ;
IF [ ABS [ #12 ] LT [ ABS [ #1 ] ] ] GOTO 2 ;
#14 = 1 ;
#13 = #1 - #12 + #13 ;
N2 G00 X #13 ;
#13 = #4 - #5 ;
END 1 ;
N3 G00 X - #1 - #5 ;
M99 ;
```

G75.1 (O751)	Groove cutting cycle
--------------	----------------------

```

G. 1 ;
#3003 = #8 OR 1 ;
G0 X #1 ;
G1 Y #2 ;
G0 Y - #2 ;
X #5 ;
1F [ #3 EQ 0 ] GOTO 1 ;
G1 X - #3 Y #4 ;
N1 G1 Y#6 ;
X - #7 ;
G0Y - #2 ;
X - #5 ;
1F [ #3 EQ 0 ] GOTO 2 ;
G1 X #3 Y #4 ;
N2 G1 Y#6 ;
X #7 ;
#3003 = #8 ;
G0Y - #2 ;
M99 ;

```

G76 (O760)	Compound thread cutting cycle
------------	-------------------------------

```

G. 1 ;
#12 = 1 ;
#13 = #9 ;
1F [ ABS [ #13 ] GE [ ABS [ #8 ] ] ] GOTO 1 ;
#16 = 1 ;
#13 = #8 ;
N1 #11 = #13 ;
1F [ ABS [ #11 ] LT [ ABS [ #4 - #5 ] ] ] GOTO 2 ;
#11 = #4 - #5 ;
#14 = 1 ;
N2 #17 = #11 ;
#18 = ROUND [ [ #4 - #11 - #5 ] * #7 ] ;
IF [ [ #18 XOR #1 ] GE 0 ] GOTO 10 ;
#18 = - #18 ;
N10 #19 = #18 ;
#10 = ROUND [ [ #11 + #5 ] * #7 ] ;
IF [ [ #10 XOR #1 ] GE 0 ] GOTO 20 ;
#10 = - #10 ;
N20 G00 X#10 ;
#20 = #10
D0 1 ;
#15 = ROUND [ #10 * #3/#1 ] ;
G00 Y #2 + #3 - #4 - #15 + #11 ;
G33 X#1 - #10 - #18 Y -#3 + #15 ;
G00 Y - #2 + #4 - #11 ;
IF [ #14 GT 0 ] GOTO 3 ;
IF [ #16 GT 0 ] GOTO 7 ;
#12 = #12 + 1 ;
#13 = ROUND [ #9 * SQRT [ #12 ] ] ;

```

```

IF [ ABS [ #13 - #11 ] GE [ ABS [ #8 ] ] ] GOTO 8 ;
#16 = 1 ;
N7 #13 = #11 + #8 ;
N8 #11 = #13 ;
IF [ ABS [ #11 ] LT [ ABS [ #4 - #5 ] ] ] GOTO 9 ;
#11 = #4 - #5 ;
#14 = 1 ;
N9 #10 = ROUND [ [ #17 - #11 ] * #7 ] ;
IF [ [ #10 XOR #1 ] GE 0 ] GOTO 6 ;
#10 = -#10 ;
N6 #10 = #10 + #20 ;
G00 X - #1 + #10 + #18 ;
IF [ #14 LT 0 ] GOTO 11 ;
#18 = 0 ;
GOTO 12 ;
N11 #18 = #19 - #10 + #20 ;
N12 END 1 ;
N3 IF [ ABS [ #6 ] LT 1 ] GOTO 5 ;
#14 = 0 ;
#13 = 0 ;
D0 2 ;
IF [ #14 GT 0 ] GOTO 5 ;
#13 = #13 + #6 ;
IF [ ABS [ #13 ] LT [ ABS [ #5 ] ] ] GOTO 4 ;
#13 = #5 ;
#14 = 1 ;
N4 G00 X #10 - #1 ;
G00 Y #2 + #3 - #4 + #13 - #15 + #11 ;
G33 X #1 - #10 Y - #3 + #15 ;
G00 Y - #2 + #4 - #13 - #11 ;
END 2 ;
N5 G00 X - #1 ;
M99 ;

```

G76.1 (O761)	2-part system simultaneous compound thread cutting cycle
--------------	--

```

G. 1 ;
N761 !L10
#12 = 1 ;
#13 = #9 ;
1F [ ABS [ #13 ] GE [ ABS [ #8 ] ] ] GOTO 1 ;
#16 = 1 ;
#13 = #8 ;
N1 #11 = #13 ;
1F [ ABS [ #11 ] LT [ ABS [ #4 - #5 ] ] ] GOTO 2 ;
#11 = #4 - #5 ;
#14 = 1 ;
N2 #17 = #11 ;
#18 = ROUND [ [ #4 - #11 - #5 ] * #7 ] ;
IF [ [ #18 XOR #1 ] GE 0 ] GOTO 10 ;
#18 = - #18 ;
N10 #19 = #18 ;
#10 = ROUND [ [ #11 + #5 ] * #7 ] ;

```

```

IF [ [#10 XOR #1 ] GE 0 ] GOTO 20 ;
#10 = - #10 ;
N20 G00 X#10 ;
#20 = #10
D0 1 ;
#15 = ROUND [ #10 * #3/#1 ] ;
G00 Y #2 + #3 - #4 - #15 + #11 ;
!L11 ;
G33 X#1 - #10 - #18 Y -#3 + #15 ;
G00 Y - #2 + #4 - #11 ;
!L12 ;
IF [ #14 GT 0 ] GOTO 3 ;
IF [ #16 GT 0 ] GOTO 7 ;
#12 = #12 + 1 ;
#13 = ROUND [ #9 * SQRT [ #12 ] ] ;
IF [ ABS [ #13 - #11 ] GE [ ABS [ #8 ] ] ] GOTO 8 ;
#16 = 1 ;
N7 #13 = #11 + #8 ;
N8 #11 = #13 ;
IF [ ABS [ #11 ] LT [ ABS [ #4 - #5 ] ] ] GOTO 9 ;
#11 = #4 - #5 ;
#14 = 1 ;
N9 #10 = ROUND [ [ #17 - #11 ] * #7 ] ;
IF [ [#10XOR#1] GE 0 ] GOTO 6 ;
#10 = -#10 ;
N6 #10 = #10 + #20 ;
G00 X - #1 + #10 + #18 ;
IF [ #14 LT 0 ] GOTO 11 ;
#18 = 0 ;
GOTO 12 ;
N11 #18 = #19 - #10 + #20 ;
N12 END 1 ;
N3 IF [ ABS [ #6 ] LT 1 ] GOTO 5 ;
#14 = 0 ;
#13 = 0 ;
D0 2 ;
IF [ #14 GOTO ] GOTO 5 ;
#13 = #13 + #6 ;
IF [ ABS [ #13 ] LT [ ABS [ #5 ] ] ] GOTO 4 ;
#13 = #5 ;
#14 = 1 ;
N4 G00 X #10 - #1 ;
G00 Y #2 + #3 - #4 + #13 - #15 + #11 ;
!L11 ;
G33 X #1 - #10 Y - #3 + #15 ;
G00 Y - #2 + #4 - #13 - #11 ;
!L12 ;
END 2 ;
N5 G00 X - #1 ;
M99 ;

```

G76.2 (O762)	2-system simultaneous compound thread cutting cycle
--------------	--

```

G. 1 ;
N762 !L10 ;
#12 = 1 ;
#13 = #9 ;
1F [ ABS [ #13 ] GE [ ABS [ #8 ] ] ] GOTO 1 ;
#16 = 1 ;
#13 = #8 ;
N1 #11 = #13 ;
1F [ ABS [ #11 ] LT [ ABS [ #4 - #5 ] ] ] GOTO 2 ;
#11 = #4 - #5 ;
#14 = 1 ;
N2 #17 = #11 ;
#18 = ROUND [ [ #4 - #11 - #5 ] * #7 ] ;
IF [ [ #18 XOR #1 ] GE 0 ] GOTO 10 ;
#18 = - #18 ;
N10 #19 = #18 ;
#10 = ROUND [ [ #11 + #5 ] * #7 ] ;
IF [ [ #10 XOR #1 ] GE 0 ] GOTO 20 ;
#10 = - #10 ;
N20 IF [# 27 NE 1 ] GOTO 21 ;
G00 X#10 ;
N21 #20 = #10 ;
#28 = 1 ;
D01 ;
#15 = ROUND [ #10 * #3/#1 ] ;
#29 = #28 MOD 2 ;
IF [ [ #27 EQ 1 ] AND [#29 EQ 0 ] ] GOTO 22 ;
IF [ [ #27 EQ 2 ] AND [#29 EQ 1 ] ] GOTO 22 ;
G00 Y #2 + #3 - #4 - #15 + #11 ;
!L11 ;
G33 X#1 - #10 - #18 Y -#3 + #15 ;
G00 Y - #2 + #4 - #11 ;
#21 = #18 ;
!L12 ;
N22 IF [ #14 GT 0 ] GOTO 3 ;
IF [ #16 GT 0 ] GOTO 7 ;
#12 = #12 + 1 ;
#13 = ROUND [ #9 * SQRT [ #12 ] ] ;
IF [ ABS [ #13 - #11 ] GE [ ABS [ #8 ] ] ] GOTO 8 ;
#16 = 1 ;
N7 #13 = #11 + #8 ;
N8 #11 = #13 ;
IF [ ABS [ #11 ] LT [ ABS [ #4 - #5 ] ] ] GOTO 9 ;
#11 = #4 - #5 ;
#14 = 1 ;
N9 #10 = ROUND [ [ #17 - #11 ] * #7 ] ;
IF [ [ #10XOR#1 ] GE 0 ] GOTO 6 ;
#10 = -#10 ;
N6 #10 = #10 + #20 ;
IF [ [ #27 EQ 1 ] AND [#29 EQ 1 ] ] GOTO 24 ;
IF [ [ #27 EQ 2 ] AND [#29 EQ 0 ] ] GOTO 24 ;
IF [ [ #27 EQ 2 ] AND [#28 EQ 1 ] ] GOTO 23 ;
G00 X - #1 + #10 + #21 ;

```

Appendix 4. Registering/Editing the Fixed Cycle Program

4.3 Standard Fixed Cycle Subprogram (For L system)

```
GOTO 24 ;
N23 G00 X#10 ;
N24 IF [ #14 LT 0 ] GOTO 11 ;
#18 = 0 ;
GOTO 12 ;
N11 #18 = #19 - #10 + #20 ;
N12 #28 = #28 + 1 ;
END 1 ;
N3 IF [ ABS [ #6 ] LT 1 ] GOTO 5 ;
#14 = 0 ;
#13 = 0 ;
D0 2 ;
IF [ #14 GT 0 ] GOTO 5 ;
#13 = #13 + #6 ;
IF [ ABS [ #13 ] LT [ ABS [ #5 ] ] ] GOTO 4 ;
#13 = #5 ;
#14 = 1 ;
N4 #29 = #28 MOD 2 ;
IF [ [ #27 EQ 1 ] AND [ #29 EQ 1 ] ] GOTO 25 ;
IF [ [ #27 EQ 2 ] AND [ #29 EQ 0 ] ] GOTO 25 ;
G00 X #10 - #1 + #21 ;
#21 = 0 ;
G00 Y #2 + #3 - #4 + #13 - #15 + #11 ;
!L11 ;
G33 X #1 - #10 Y - #3 + #15 ;
G00 Y - #2 + #4 - #13 - #11 ;
!L12 ;
N25 #28 = #28 + 1 ;
END 2 ;
N5 G00 X - #1 ;
M99 ;
%
```

G77 (O770)	Longitudinal cutting cycle
------------	----------------------------

```
G. 1 ;
1F [ [ #1 EQ 0 ] OR [ #2 EQ 0 ] ] GOTO 1 ;
Y #2 + #7 ;
G1 X #1 Y - #7 ;
Y - #2 ;
G0 X - #1 ;
N1 M99 ;
```

G78 (O780)	Thread cutting cycle
------------	----------------------

```
G. 1 ;
1F [ [ #1 EQ 0 ] OR [ #2 EQ 0 ] ] GOTO 1 ;
Y #2 + #7 ;
G33 X #1 Y - #7 F #9 E #10 ;
G0 Y - #2 ;
X - #1 ;
N1 M99 ;
```

Appendix 4. Registering/Editing the Fixed Cycle Program

4.3 Standard Fixed Cycle Subprogram (For L system)

G79 (O790)	End face cutoff cycle
------------	-----------------------

```
G. 1 ;  
1F [ [ #1 EQ 0 ] OR [ #2 EQ 0 ] ] GOTO 1 ;  
X #1 + #7 ;  
G1 X - #7 Y #2 ;  
X - #1 ;  
G0 Y - #2 ;  
N1 M99 ;
```

G83 G87 (O830)	Deep hole drilling cycle B
-------------------	----------------------------

```
G. 1 ;  
1F [ #30 ] GOTO 2 ;  
M #24 ;  
#29 = #11 #28 = 0 ;  
Z #2 ;  
#2 = ##5 #3003 = #8 OR 1 ;  
D0 1 ;  
#28 = #28 - #11 #26 = - #28 - #29 ;  
Z #26 ;  
IF [ ABS [ #28 ] GE [ ABS [ #3 ] ] ] GOTO 1 ;  
G1 Z #29 ;  
G0 Z #28 ;  
G29 = #11 + #14 ;  
END 1 ;  
N1 G1 Z #3 - #26 ;  
G4 P #4 ;  
#3003 = #8 ;  
G0Z - #3 - #2 ;  
IF [ #24 EQ #0 ] GOTO 2 ;  
M #24 + 1 ;  
G4 P #21 ;  
N2 M99 ;
```

Appendix 4. Registering/Editing the Fixed Cycle Program

4.3 Standard Fixed Cycle Subprogram (For L system)

G83 G87	(O831)	Deep hole drilling cycle A
------------	--------	----------------------------

```
G. 1 ;
1F [ #30 ] GOTO 2 ;
M #24 ;
#29 = #0 #28 = #11 ;
Z #2 ;

#2 = ##5 #3003 = #8 OR 1 ;
D0 1 ;
#29 = #29 + #11 ;
IF [ ABS [ #29 ] GE [ ABS [ #3 ] ] ] GOTO 1 ;
G1 Z #28 ;
G0 Z - #14 ;
#28 = #11 + #14 ;
END 1 ;
N1 G1 Z #3 - #29 + #28 ;
G4 P #4 ;
#3003 = #8 ;
G0Z - #3 - #2 ;
IF [#24 EQ #0 ] GOTO 2 ;
M #24 + 1 ;
G4 P #21 ;
N2 M99 ;
```

G83.2	(O832)	Deep hole drilling cycle 2
-------	--------	----------------------------

```
G. 1 ;
1F [ #30 ] GOTO 3 ;
#3003 = #8 OR 1 ;
#29 = #12 #28 = 0 ;
G0 Z #2 ;
IF [ #12 NE #0 ] GOTO 1 ;
IF [ #11 EQ #0 ] GOTO 2 ;
N1 #28 = #28 - #12 #26 = - #28 - #29 ;
IF [ ABS [ #28 ] GE [ ABS [ #3 ] ] ] GOTO 2 ;
G1 Z #12 ;
G4 P #4 ;
G0 Z #28 - #2 ;
G4P # 13 ;
G29 = #11 + #15 ;
D0 1 ;
#28 = #28 - #11 #26 = - #28 - #29 ;
G0 Z #26 + #2 ;
IF [ ABS [ #28 ] GE [ ABS [ #3 ] ] ] GOTO 2 ;
G1 Z #29 ;
G4 P #4 ;
G0 Z #28 - #2 ;
G4 P # 13 ;
END 1 ;
N2 G1 Z #3 - #26 ;
G4 P #4 ;
#3003 = #8 ;
G0Z - #3 - #2 ;
N3 M99 ;
```

Appendix 4. Registering/Editing the Fixed Cycle Program

4.3 Standard Fixed Cycle Subprogram (For L system)

G84 G88	(O840)	Tap cycle
------------	--------	-----------

```
G. 1 ;  
1F [ #30 ] GOTO 2 ;  
M #24 ;  
Z #2 ;  
#2 = ##5 #3003 = #8 OR 1 #3004 = #9 OR 3 ;  
G1 Z #3 ;  
G4 P #4 ;  
M4 ;  
#3900 = 1 ;  
G1 Z - #3 ;  
#3004 = #9 ;  
M3 ;  
#3003 = #8 ;  
IF [#24 EQ #0 ] GOTO 1 ;  
M #24 + 1 ;  
G4 P #21 ;  
N1 G0 Z - #2 ;  
N2 M99 ;
```

G85 G89	(O850)	Boring cycle
------------	--------	--------------

```
G. 1 ;  
1F [ #30 ] GOTO 2 ;  
M #24 ;  
Z #2 ;  
#2 = ##5 #3003 = #8 OR 1 ;  
G1 Z #3 ;  
G4 P #4 ;  
#3003 = #8 ;  
Z - #3 F #23 ;  
F #22 ;  
IF [#24 EQ #0 ] GOTO 1 ;  
M #24 + 1 ;  
G4 P #21 ;  
N1 G0 Z - #2 ;  
N2 M99 ;
```

4.4 Standard Fixed Cycle Subprogram (For M system)

G81 (O810)	Drill, spot drill
------------	-------------------

```
G. 1 ;
1F [#30] GOTO1 ;
Z#2 G#6 H#7 ;
#2 = ##5
#3003 = #8 OR 1 ;
G1 Z#3 ;
#3003 = #8 ;
G0 Z - #3 - #2 ;
N1 M99%
```

Fixed cycle block 1 movement command
Check for fixed cycle invalidity.

Inhibit single block stop.

Return.

G82 (O820)	Drill, counter boring
------------	-----------------------

```
G. 1 ;
1F [#30] GOTO1 ;
Z#2 G#6 H#7 ;
#2 = ##5
#3003 = #8 OR 1 ;
G1 Z#3 ;
G4 P#4 ;
#3003 = #8 ;
G0 Z - #3 - #2 ;
N1 M99%
```

Fixed cycle block 1 movement command
Check for fixed cycle invalidity.

Inhibit single block stop.

Dwell.

Return.

G83 (O830)	Deep hole drill cycle
------------	-----------------------

```
G. 1 ;
1F [#30] GOTO2 ;
#29 = #11
#28 = 0
Z#2 G#6 H#7 ;
#2 = ##5
#3003 = #8 OR 1 ;
DO 1 ;
#28 = #28 - #11
#26 = - #28 - #29
Z#26 ;
IF [ABS [#28] GE [ABS [#3] ] ] GOTO 1 ;

G1 Z#29 ;
```

Fixed cycle block 1 movement command
Check for fixed cycle invalidity.
Define the cutting amount.
Initialize the return amount (total cutting amount).

Inhibit single block stop.

Define the return amount of the next block.

Calculate the feed amount.

Feed.

Does the total cutting amount (return amount) exceed the cut amount?

Cutting feed

Appendix 4. Registering/Editing the Fixed Cycle Program

4.4 Standard Fixed Cycle Subprogram (For M system)

```
G0 Z#28 ;
#29 = #11 + #14 ;
END1 ;
N1 G1 Z#3 - #26 ;
#3003 = #8 ;
G0 Z - #3 - #2 ;
N2 M99%
```

Return.

Define the cutting amount for block 2 and after.

Cutting feed

Return.

G84 (O840)	Tap cycle
------------	-----------

```
G. 1 ;
1F [#30] GOTO1 ;
Z#2 G#6 H#7 ;
#2 = ##5
#3003 = #8 OR 1 ;
#3004 = #9 OR 3 ;
G1 Z#3 ;
G4 P#4 ;
M4 ;
#3900 = 1 ;
G1 Z - #3 ;
#3004 = #9 ;
G4 P#4 ;
M3 ;
#3003 = #8 ;
G0 Z = #2 ;
N1 M99%
```

Fixed cycle block 1 movement command

Check for fixed cycle invalidity.

Inhibit single block stop.

Invalidate feed hold/override.

Dwell.

Rotate the spindle reversely.

Dwell.

Rotate the spindle forward.

Return.

G85 (O850)	Boring 1
------------	----------

```
G. 1 ;
1F [#30] GOTO1 ;
Z#2 G#6 H#7 ;
#2 = ##5
#3003 = #8 OR 1 ;
G1 Z#3 ;
#3003 = #8 ;
Z - #3 ;
G0 Z - #2 ;
N1 M99%
```

Fixed cycle block 1 movement command

Check for fixed cycle invalidity.

Inhibit single block stop.

Return.

Appendix 4. Registering/Editing the Fixed Cycle Program

4.4 Standard Fixed Cycle Subprogram (For M system)

G86 (O860)	Boring 2
------------	----------

```
G. 1 ;  
1F [#30] GOTO1 ;  
Z#2 G#6 H#7 ;  
#2 = ##5  
#3003 = #8 OR 1 ;  
G1 Z#3 ;  
G4 P#4 ;  
M5 ;  
G0 Z - #3 - #2 ;  
#3003 = #8 ;  
M3 ;  
N1 M99%
```

Fixed cycle block 1 movement command
Check for fixed cycle invalidity.

Inhibit single block stop.

Dwell.
Stop the spindle.
Return.

Rotate the spindle forward.

G87 (O870)	Back boring
------------	-------------

```
G. 1 ;  
1F [#30] GOTO1 ;  
#3003 = #8 OR 1 ;  
M19 ;  
X#12 Y#13 ;  
#3003 = #8 ;  
Z#2 G#6 H#7 ;  
#3003 = #8 OR 1 ;  
G1 X - #12 Y - #13 ;  
#3003 = #8 ;  
M3 ;  
#3003 = #8 OR 1 ;  
Z#3 ;  
M19 ;  
G0 X#12 Y#13 ;  
Z - #2 - #3 ;  
#3003 = #8 ;  
X - #12 Y - #13 ;  
M3 ;  
N1 M99%
```

Fixed cycle block 1 movement command
Check for fixed cycle invalidity.

Inhibit single block stop.
Orient the spindle.

Cancel single block stop inhibition.

Inhibit single block stop.

Cancel single block stop inhibition.
Rotate the spindle forward.
Inhibit single block stop.

Orient the spindle.

Shift.

G87 is not affected by the G98 or G99 modal.

Cancel single block stop inhibition.

Shift.

Rotate the spindle forward.

Appendix 4. Registering/Editing the Fixed Cycle Program

4.4 Standard Fixed Cycle Subprogram (For M system)

G88 (O880)	Boring 3
------------	----------

```
G. 1 ;
1F [#30] GOTO1 ;
Z#2 G#6 H#7 ;
#2 = ##5
#3003 = #8 OR 1 ;
G1 Z#3 ;
G4 P#4 ;
#3003 = #8 ;
M5 ;
#3003 = #8 OR 1 ;
G0 Z - #3 - #2 ;
#3003 = #8 ;
M3 ;
N1 M99 ;
```

Fixed cycle block 1 movement command
Check for fixed cycle invalidity.

Inhibit single block stop.

Dwell.

Cancel single block stop inhibition.

Rotate the spindle forward

Inhibit single block stop.

Return.

Cancel single block stop inhibition.

Rotate the spindle forward.

G89 (O890)	Boring 4
------------	----------

```
G. 1 ;
1F [#30] GOTO1 ;
Z#2 G#6 H#7 ;
#2 = ##5
#3003 = #8 OR 1 ;
G1 Z#3 ;
G4 P#4 ;
#3003 = #8
Z - #3 ;
G0 Z - #2 ;
N1 M99%
```

Fixed cycle block 1 movement command
Check for fixed cycle invalidity.

Inhibit single block stop.

Dwell.

Return.

G73 (O831)	Step cycle
------------	------------

```
G. 1 ;
1F [#30] GOTO2 ;
#29 = 0
#28 = #11 ;
Z#2 G#6 H#7 ;
#2 = ##5
#3003 = #8 OR 1 ;
DO 1 ;
#29 = #29 + #11 ;
IF [ABS [#29] GE [ABS [#3] ] ] GOTO 1 ;
```

Fixed cycle block 1 movement command
Check for fixed cycle invalidity.

Initialize the total cutting amount.

Define the cutting amount.

Single block stop command

Increment the total cutting amount counter.

Does the total cutting amount exceed the cut amount Z?

Appendix 4. Registering/Editing the Fixed Cycle Program

4.4 Standard Fixed Cycle Subprogram (For M system)

G1 Z#28 ;	Cutting feed
G4 P#4 ;	Dwell.
G0 Z - #14 ;	Return.
#28 = #11 + #14 ;	Define the cutting amount for block 2 and after.
END1 ;	
N1 G1 Z#3 - #29 + #28 ;	Cutting feed
G4 P#4 ;	Dwell.
#3003 = #8 ;	
G0Z - #3 - #2 ;	Return.
N2 M99%	

G74 (O841)	Reverse tap cycle
------------	-------------------

G. 1 ;	Fixed cycle block 1 movement command
1F [#30] GOTO1 ;	Check for fixed cycle invalidity.
Z#2 G#6 H#7 ;	
#2 = ##5	
#3003 = #8 OR 1 ;	Inhibit single block stop.
#3004 = #9 OR 3 ;	Invalidate feed hold/override.
G1 Z#3 ;	
G4 P#4 ;	Dwell.
M3 ;	Rotate the spindle forward.
#3900 = 1 ;	
Z - #3 ;	
#3004 = #9 ;	
G4 P#4 ;	Dwell.
M4 ;	Rotate the spindle reversely.
#3003 = #8 ;	
G0 Z - #2 ;	
N1 M99%	Return.

Appendix 4. Registering/Editing the Fixed Cycle Program

4.4 Standard Fixed Cycle Subprogram (For M system)

G75 (O750)	Circle cutting cycle
------------	----------------------

```

G. 1 ;
IF [#30] GOTO 1 ;
#28=#18;
#2 = ##5
IF[#28GE0]GOTO2;
#27=3#28=-#28;
GOTO3;
N2 #27=2;
N3 #26=[#[16000+#4]+#[17000+#4]]*#99;
IF[#26GE#28]GOTO1;
Z#2G#6H#7;
#2=##5#3003=#8OR1;
G1Z#3;
#28=#28-#26#29=#28/2;
G#27X-#28I-#29;
I#28P1;
X#28I#29;
#3003=#8;
G0Z-#3-#2;
N1 M99%
```

Fixed cycle block 1 movement command.
Check for fixed cycle invalidity.

Decide the circular direction.
G03 circular

G02 circular

Check compensation amount

Inner circumference half lap
Outer circumference one lap
Inner circumference half lap

Return.

G76 (O861)	Fine boring
------------	-------------

```

G. 1 ;
1F [#30] GOTO1 ;
Z#2 G#6 H#7 ;
#2 = ##5
#3003 = #8 OR 1 ;
G1 Z#3 ;
M19 ;
X#12 Y#13 ;
G0 Z - #3 - #2 ;
#3003 = #8 ;
X - #12 Y - #13 ;
M3 ;
N1 M99%
```

Fixed cycle block 1 movement command
Check for fixed cycle invalidity.

Single block stop command

Orient the spindle.
Shift.
Return.

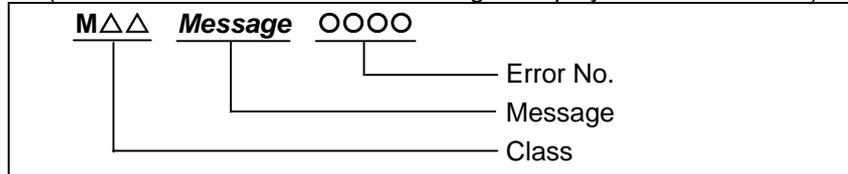
Shift.
Rotate the spindle forward.

Appendix 6. Alarms

6.1 List of Alarms

6.1.1 Operation Alarms

(The bold characters are the messages displayed on the screen.)



(1) Class: M01 Operation error

Alarms occurring due to incorrect operation by the operator during NC operation and those by machine trouble are displayed.

Error No.	Details	Remedy
0001	Dog overrun When returning to the reference position, the near-point detection limit switch did not stop over the dog, but overran the dog.	<ul style="list-style-type: none"> • Increase the length of the near-point dog. • Reduce the reference position return speed.
0002	Some ax does not pass Z phase One of the axes did not pass the Z-phase during the initial reference position return after the power was turned ON.	<ul style="list-style-type: none"> • Move the detector one rotation or more in the opposite direction of the reference position, and repeat reference position return.
0003	R-pnt direction illegal When manually returning to the reference position, the return direction differs from the axis movement direction selected with the AXIS SELECTION key.	<ul style="list-style-type: none"> • The selection of the AXIS SELECTION key's +/- direction is incorrect. The error is canceled by feeding the axis in the correct direction.
0004	External interlock axis exists The external interlock function has activated (the input signal is "OFF") and one of the axes has entered the interlock state.	<ul style="list-style-type: none"> • As the interlock function has activated, release it before resuming operation. • Check the sequence on the machine side. • Check for broken wires in the interlock signal line.
0005	Internal interlock axis exists The internal interlock state has been entered. The absolute position detector axis has been removed. A command for the manual/automatic simultaneous valid axis was issued from the automatic mode. The manual speed command was issued while the tool length measurement 1 signal is ON.	<ul style="list-style-type: none"> • The servo OFF function is valid, so release it first. • An axis that can be removed has been issued, so perform the correct operations. • The command is issued in the same direction as the direction where manual skip turned ON, so perform the correct operations. • During the manual/automatic simultaneous mode, the axis commanded in the automatic mode became the manual operation axis. Turn OFF the manual/automatic valid signal for the commanded axis. • Turn ON the power again, and perform absolute position initialization. • Turn OFF the tool length measurement 1 signal to start the program by the manual speed command.
0006	H/W stroke end axis exists The stroke end function has activated (the input signal is "OFF") and one of the axes is in the stroke end status.	<ul style="list-style-type: none"> • Move the machine manually. • Check for broken wires in the stroke end signal wire. • Check for trouble in the limit switch.

Error No.	Details	Remedy
0007	S/W stroke end axis exists The stored stroke limit I, II, IIB or IB function has activated.	<ul style="list-style-type: none"> Move it manually. If the stored stroke limit in the parameter is incorrectly set, correct it.
0008	Chuck/tailstock stroke end ax The chuck/tail-stock barrier function turned ON, and an axis entered the stroke end state.	<ul style="list-style-type: none"> Reset the alarm with reset, and move the machine in the reverse direction.
0009	Ref point return No. invalid Return to the No. 2 reference position was performed before return to the No. 1 reference position was completed.	<ul style="list-style-type: none"> Execute No. 1 reference position return.
0019	Sensor signal illegal ON The sensor signal was already ON when the tool measurement mode (TLM) signal was validated. The sensor signal turned ON when there was no axis movement after the tool measurement mode (TLM) signal was validated. The sensor signal turned ON at a position within 100 μ m from the final entry start position.	<ul style="list-style-type: none"> Turn the tool measurement mode signal input OFF, and move the axis in a safe direction. The operation alarm will turn OFF even when the sensor signal is turned OFF. <p>(Note) When the tool measurement mode signal input is turned OFF, the axis can be moved in either direction. Pay attention to the movement direction.</p>
0020	Ref point retract invalid Return to the reference position was performed before the coordinates had not been established.	<ul style="list-style-type: none"> Execute reference position return
0021	Tool ofs invld after R-pnt Reference position return was performed during tool retract return, and therefore the tool compensation amount became invalid after reference position return was completed.	<ul style="list-style-type: none"> The error is cleared if the operation mode is changed to other than reference position return before the axis performs reference position return. The error is cleared when reference position return is completed. The error is cleared if reset 1 is input or the emergency stop button is pushed.
0024	R-pnt ret invld at abs pos alm A zero point return signal was input during an absolute position detection alarm.	<ul style="list-style-type: none"> Reset the absolute position detection alarm, and then perform zero point return.
0025	R-pnt ret invld at zero pt ini A zero point return signal was input during zero point initialization of the absolute position detection system.	<ul style="list-style-type: none"> Complete zero point initialization, and then perform zero point return.
0030	Now skip on The skip signal remains input when the skip return operation changed to the measurement operation.	<ul style="list-style-type: none"> Increase the skip return amount.
0031	No skip Even though 1st skip was to the correct position, the 2nd skip could not be found.	<ul style="list-style-type: none"> Check whether the measurement target has moved.
0050	Chopping axis R-pnt incomplete The chopping axis has not completed zero point return before entering the chopping mode. All axes interlock will be applied.	<ul style="list-style-type: none"> Reset or turn the chopping signal OFF, and then carry out zero point return.

Error No.	Details	Remedy
0051	<p>Synchronous error excessive</p> <p>The synchronization error of the master and slave axes exceeded the allowable value under synchronous control.</p> <p>A deviation exceeding the synchronization error limit value was found with the synchronization deviation detection.</p>	<ul style="list-style-type: none"> • Select the correction mode and move one of the axes in the direction in which the errors are reduced. • Increase the allowable value or reset it to 0 (check disabled). • When using simple C-axis synchronous control, set the contents of the R435 register to 0. • Check the parameter (#2024 synerr).
0053	<p>No spindle select signal</p> <p>Synchronous tapping command was issued when the spindle select signals (SWS) for all spindles were OFF in the multiple-spindle control II.</p>	<ul style="list-style-type: none"> • Turn ON the spindle select signal (SWS) responding to the tapping spindle before performing the synchronous tapping command.
0054	<p>No spindle serial connection</p> <p>Synchronous tapping command was issued when the spindle that the spindle select signal (SWS) was ON was not serially connected in the multiple-spindle control II.</p>	<ul style="list-style-type: none"> • Make sure the spindle select signal (SWS) for the responding spindle is ON. • When issuing a command, consider the machine construction.
0055	<p>Spindle fwd/rvs run para err</p> <p>Asynchronous tapping command was issued when M code of the spindle frd/rvs run command set by the parameter "#3028 sprcmm" was one of the followings in the multiple-spindle control II.</p> <ul style="list-style-type: none"> • One of M0, M1, M2, M30, M98, M99, M198 • M code No. that commands macro interrupt signal valid/invalid 	<ul style="list-style-type: none"> • Change the value of the parameter #3028 sprcmm.
0056	<p>Tap pitch/thread number error</p> <p>The command of the pitch/thread number is not correct in the synchronous tapping command of the multiple-spindle control II.</p> <p>The pitch is too small for the spindle rotation speed.</p> <p>Thread number is too large for the spindle rotation speed.</p>	<ul style="list-style-type: none"> • Check the pitch/thread number and rotation speed of the tapping spindle.
0060	<p>Handle ratio too large</p> <p>Handle ratio is too large for the rapid traverse rate (or external deceleration speed when external deceleration is valid).</p>	<ul style="list-style-type: none"> • Set a smaller ratio.
0065	<p>R-pos offset value illegal</p> <p>At the start of reference position initial setting, setting of reference position offset value (#2034 rpfps) is other than 0.</p>	<ul style="list-style-type: none"> • Set the reference position offset value (#2034 rpfps) to 0, then turn the power ON again to perform reference position initial setting.
0066	<p>R-pos scan distance exceeded</p> <p>Reference position could not be established within the maximum scan distance.</p>	<ul style="list-style-type: none"> • Check the scale to see if it has dirt or damage. • Check if the servo drive unit supports this function.

Error No.	Details	Remedy
0101	No operation mode	<ul style="list-style-type: none"> • Check for a broken wire in the input mode signal wire. • Check for trouble in the mode selector switch. • Check the sequence program.
0102	Cutting override zero The "cutting feed override" switch on the machine operation panel is set to zero. The override was set to "0" during a single block stop.	<ul style="list-style-type: none"> • Set the "cutting feed override" switch to a value other than zero to clear the error. • When the "cutting feed override" switch is set to a value other than zero, check for a short circuit in the signal wire. • Check the sequence program.
0103	External feed rate zero "The manual feed speed" switch on the machine operation panel is set to zero when the machine is in the jog mode or automatic dry run mode. The "Manual feedrate B speed" is set to zero during the jog mode when manual feedrate B is valid. The "each axis manual feedrate B speed" is set to zero during the jog mode when each axis manual feedrate B is valid.	<ul style="list-style-type: none"> • Set "the manual feed speed" switch to a value other than zero to release the error. • If "the manual feed speed" switch is set to a value other than zero, check for a short circuit in the signal wire. • Check the sequence program.
0104	F 1-digit feed rate zero The F1-digit feedrate is set to zero when the F1-digit feed command is being executed.	<ul style="list-style-type: none"> • Set the F1-digit feedrate on the setup parameter screen.
0105	Spindle stop The spindle stopped during the synchronous feed command.	<ul style="list-style-type: none"> • Rotate the spindle. • If the workpiece is not being cut, start dry run. • Check for a broken wire in the spindle encoder cable. • Check the connections for the spindle encoder connectors. • Check the spindle encoder pulse. • Reconsider the program. (Command, address)
0106	Handle feed ax No. illegal An axis not found in the specifications was designated for handle feed or the handle feed axis was not selected.	<ul style="list-style-type: none"> • Check for broken wires in the handle feed axis selection signal wire. • Check the sequence program. • Check the No. of axes listed in the specifications.
0107	Spindle rotation speed over The spindle rotation speed exceeded the axis clamp speed during the thread cutting command.	<ul style="list-style-type: none"> • Lower the commanded spindle rotation speed.
0108	Fixed pnt mode feed ax illegal An axis not found in the specifications was designated for the fixed point mode feed or the fixed point mode feedrate is illegal.	<ul style="list-style-type: none"> • Check for broken wires in the fixed mode feed axis selection signal wire and fixed point mode feedrate wire. • Check the fixed point mode feed specifications.
0109	Block start interlock An interlock signal that locks the start of the block has been input.	<ul style="list-style-type: none"> • Check the sequence program.
0110	Cutting block start interlock An interlock signal that locks the start of the cutting block has been input.	<ul style="list-style-type: none"> • Check the sequence program.

Error No.	Details	Remedy
0111	Restart switch ON The restart switch was turned ON before the restart search was completed, and the manual mode was selected.	<ul style="list-style-type: none"> • Search the block to be restarted. • Turn OFF the restart switch.
0112	Program check mode The automatic start button was pressed during program check or in program check mode.	<ul style="list-style-type: none"> • Press the reset button to cancel the program check mode.
0113	Auto start in buffer correct The automatic start button was pressed during buffer correction.	<ul style="list-style-type: none"> • Press the automatic start button after buffer correction is completed.
0115	In reset process The automatic start button was pressed during resetting or tape rewinding.	<ul style="list-style-type: none"> • When rewinding the tape, wait for the winding to end, or press the reset button to stop the winding, and then press the automatic start button. • During resetting, wait for resetting to end, and then press the automatic start button.
0117	Playback not possible The playback switch was turned ON during editing.	<ul style="list-style-type: none"> • During editing, cancel the function by pressing the input or previous screen key, and then turn ON the playback switch.
0118	Turn stop in normal line ctrl The turning angle at the block joint exceeded the limit during normal line control. Normal line control type I The normal line control axis turning speed (#1523 C_feed) has not been set. Normal line control type II When turning in the inside of the arc, the parameter "#8041 C-rot. R" setting value is larger than the arc radius.	<ul style="list-style-type: none"> • Check the program. • Set the normal line control axis turning speed. (Parameter "#1523 C_feed") • Set the C axis turning diameter smaller than the arc radius, or check the setting value of the C axis turning diameter. (Parameter "#8041 C rot. R")
0119	Reverse run impossible Any of the following conditions are occurring. a) There is no block to run backward b) Eight blocks without a travel command continued	<ul style="list-style-type: none"> • a) Release with forward run. • b) Release with reset.
0120	In synchronous correction mode The synchronous correction mode switch was pressed in a non-handle mode.	<ul style="list-style-type: none"> • Select the handle or manual feed mode. • Turn OFF the correction mode switch.
0121	No synchronous control option The synchronous control system (register R2589) was set with no synchronous control option.	<ul style="list-style-type: none"> • Set 0 in register R2589.
0123	Computer link B not possible The cycle start was attempted before resetting was completed. An attempt was made to perform computer link B operation at the second part system and following in a multi-part system.	<ul style="list-style-type: none"> • Perform the cycle start after resetting is completed. • Set 0 in "#8109 HOST LINK", and then set 1 again before performing the cycle start. • The computer link B operation cannot be performed at the second part system and following in a multi-part system.
0124	X/Z axes simultaneous prohibit The basic axis corresponding to the inclined axis was started simultaneously in the manual mode while the inclined axis control was valid.	<ul style="list-style-type: none"> • Turn the inclined axis and basic axis start OFF for both axes. (This also applied for manual/automatic simultaneous start.) • Invalidate the basic axis compensation, or command one axis at a time.

Error No.	Details	Remedy
0125	Rapid override zero The "rapid traverse override" switch on the machine operation panel is set to zero.	<ul style="list-style-type: none"> • Set the "rapid traverse override" switch to a value other than zero to clear the error. • When the "rapid traverse override" switch is set to a value other than zero, check for a short circuit in the signal wire. • Check the sequence program.
0126	Program restart machine lock Machine lock was applied on the return axis while manually returning to the restart position.	<ul style="list-style-type: none"> • Release the machine lock before resuming operations.
0127	Rot axis parameter error The orthogonal coordinate axis name does not exist. The rotary axis name does not exist. The orthogonal coordinate axis name is duplicated. The number of axes that were selected to change tool length compensation along the tool axis amount exceeds the maximum number of axes. The orthogonal coordinate axis name is that of the rotary axis name.	<ul style="list-style-type: none"> • Review the rotational axis configuration parameters.
0128	Restart pos return incomplete Automatic return was performed with an axis whose return to the restart position was not complete.	<ul style="list-style-type: none"> • Perform restart position return manually. • Validate the parameter "automatic return by program restart" (#1302 AutoRP), then execute automatic start.
0150	Chopping override zero The override became "0" while performing the chopping operation.	<ul style="list-style-type: none"> • Check the chopping override (R2530). • Check the rapid traverse override (R2502).
0151	Command axis chopping axis A chopping axis movement command was issued from the program during the chopping mode. (This alarm will not occur when the movement amount is commanded as 0.) (All axes interlock state will be applied.)	<ul style="list-style-type: none"> • Reset, or turn OFF the chopping signal. When the chopping signal is turned OFF, the axis will return to the reference position, and then the program movement command will be executed.
0153	Bottom dead center pos. zero The bottom dead center position is set to the same position as the upper dead center position.	<ul style="list-style-type: none"> • Correctly set the bottom dead center position.
0154	Chopping disable for handle ax Chopping was started when the chopping axis was selected as the handle axis.	<ul style="list-style-type: none"> • Select an axis other than the chopping axis as the handle axis, or start chopping after changing the mode to another mode.
0160	No speed set out of soft limit Returned from the outside of the soft limit range for the axis with no maximum speed set for the outside of the soft limit range.	<ul style="list-style-type: none"> • Set the maximum speed for the outside of the soft limit range. (Parameter "#2021 out_f") • Change the soft limit range. (Parameter "#2013 OT-" "#2014 OT+")

Error No.	Details	Remedy
0166	<p>Aux axis changeover error One of the following attempts was made on an axis that is switchable between NC axis and auxiliary axis.</p> <ul style="list-style-type: none"> • A command was issued to an auxiliary axis from machining program. • When there were more than one NC axis having a same name, a command was issued to those axes from machining program. • NC axis control select signal was turned OFF while the NC axis was in motion. • NC axis control select signal was turned ON while the auxiliary axis was in motion. 	<ul style="list-style-type: none"> • If you wish to issue a command to the axis from machining program, turn ON the NC axis control select signal so as to set the axis as an NC axis. • When more than one axis have a same name, let only one of the axes work as an NC axis. • Do not change NC axis control select signal while the axis is in motion.
0170	<p>Ill. op during T tip control An attempt was made to perform an incorrect operation during tool tip center control.</p>	<ul style="list-style-type: none"> • Change to the previous operation mode and reboot.
1005	<p>G114.n command illegal An attempt was made to execute G114.n during execution of G114.n. G51.2 was commanded when the G51.2 spindle-spindle polygon machining mode was already entered with a separate part system.</p>	<ul style="list-style-type: none"> • Cancel with G113. • Issue the spindle synchronous cancel signal (Y18B8: SPSYC). • Cancel with G50.2. • Cancel with the spindle-spindle polygon cancel signal (YCD1).
1007	<p>Spindle in-use by synchro tap The spindle is being used in synchronized tapping.</p>	<ul style="list-style-type: none"> • Cancel synchronized tapping.
1026	<p>SP-C ax ctrl runs independtly C axis mode command was issued for polygon machining spindle. C axis mode command was issued for synchronized tapping spindle. Polygon command was issued for synchronized tapping spindle. Spindle is being used as spindle/C axis.</p>	<ul style="list-style-type: none"> • Cancel the C axis command. • Cancel the polygon machining command. • Cancel the C axis with servo OFF.
1030	<p>Synchronization mismatch Different M codes were commanded in the two part systems as the synchronization M codes. Synchronization with the "!" code was commanded in another part system during M code synchronization. Synchronization with the M code was commanded in another part system during synchronization with the "!" code.</p>	<ul style="list-style-type: none"> • Correct the program so that the M codes match. • Correct the program so that the same synchronization codes are commanded.
1031	<p>Multiple C axes select invalid The C axis selection signal was changed when multiple C axes could not be selected. An axis that cannot be controlled as the multiple C axes selection was selected.</p>	<ul style="list-style-type: none"> • Check and correct the parameters and program.

Error No.	Details	Remedy
1032	Tap retract Sp select illegal Tap return was executed when a different spindle was selected. Cutting feed will wait until synchronization is completed.	<ul style="list-style-type: none"> • Select the spindle for which tap cycle was halted before the tap return signal was turned ON.
1033	Sp-Sp polygon cut interlock Cutting feed will wait until synchronization is completed.	<ul style="list-style-type: none"> • Wait for synchronization to end.
1034	Mixed sync ctrl prmtr illegal Mixed synchronization control exceeding the number of control axes was attempted. Mixed synchronization control with duplicated axis addresses was attempted.	<ul style="list-style-type: none"> • Check the parameter settings for mixed synchronization control.
1035	Mixed sync ctrl disable modal Mixed synchronization was commanded for a part system in which mixed synchronization control is disabled as shown below. <ul style="list-style-type: none"> • During nose R compensation mode • During pole coordinate interpolation mode • During cylindrical interpolation mode • During balance cut mode • During fixed cycle machining mode • During facing turret mirror image 	<ul style="list-style-type: none"> • Check the program.
1036	Synchro ctrl setting disable The synchronous control operation method selection (R2589 register) was set when the mode was not the C axis mode. The synchronous control operation method selection (R2589 register) was set in the zero point not set state. Mirror image disable state The external mirror image or parameter mirror image was commanded during facing turret mirror image.	<ul style="list-style-type: none"> • Set the R2589 register to 0. • Check the program and parameters.
1037	Synchro start/cancel disable Synchronous control was started or canceled when synchronous control could not be started or canceled.	<ul style="list-style-type: none"> • Check the program and parameters.
1038	Move cmdn invld to synchro ax A movement command was issued to a synchronous axis in synchronous control.	<ul style="list-style-type: none"> • Check the program.
1106	Sp synchro phase calc illegal The spindle synchronization phase alignment command was issued while the spindle synchronization phase calculation request signal was ON.	<ul style="list-style-type: none"> • Check the program. • Check the sequence program.

(2) Class: M90 Message: Parameter set mode

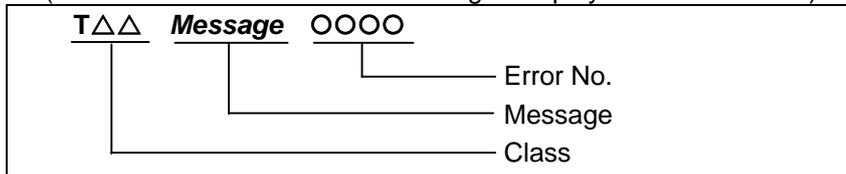
M90 Messages output when the setup parameter lock function is enabled are displayed.

Error No.	Details	Remedy
-	Setup parameter lock released The setup parameter lock is released. Automatic start is disabled when setup parameters can be set.	Refer to the manual issued by the machine tool builder.

6.1.2 Stop Codes

These codes indicate a status that caused the controller to stop for some reason.

(The bold characters are the messages displayed on the screen.)



(1) Class: T01 Cycle start prohibit

This indicates the state where automatic operation cannot be started when attempting to start it from the stop state.

Error No.	Details	Remedy
0101	Axis in motion Automatic start is not possible as one of the axes is moving.	<ul style="list-style-type: none"> Try automatic start again after all axes have stopped.
0102	NC not ready Automatic start is not possible as the NC is not ready.	<ul style="list-style-type: none"> Another alarm has occurred. Check the details and remedy.
0103	Reset signal ON Automatic start is not possible as the reset signal has been input.	<ul style="list-style-type: none"> Turn OFF the reset input signal. Check that the reset switch is not ON constantly due to trouble. Check the sequence program.
0104	Auto operation pause signal ON The FEED HOLD switch on the machine operation panel is ON (valid).	<ul style="list-style-type: none"> Check the FEED HOLD switch. The feed hold switch is the B contact. Check for broken wires in the feed hold signal wire. Check the sequence program.
0105	H/W stroke end axis exists Automatic start is not possible as one of the axes is at the stroke end.	<ul style="list-style-type: none"> If one of the axis' ends is at the stroke end, move the axis manually. Check for broken wire in the stroke end signal wire. Check for trouble in the stroke end limit switch.
0106	S/W stroke end axis exists Automatic start is not possible as one of the axes is at the stored stroke limit.	<ul style="list-style-type: none"> Move the axis manually. If an axis is not at the end, check the parameter details.
0107	No operation mode The operation mode has not been selected.	<ul style="list-style-type: none"> Select the automatic operation mode. Check for broken wires in the automatic operation mode (memory, tape, MDI) signal wire.
0108	Operation mode duplicated Two or more automatic operation modes are selected.	<ul style="list-style-type: none"> Check for a short circuit in the mode selection signal wire (memory, tape, MDI). Check for trouble in the switch. Check the sequence program.
0109	Operation mode changed The automatic operation mode changed to another automatic operation mode.	<ul style="list-style-type: none"> Return to the original automatic operation mode, and start automatic start.

Error No.	Details	Remedy
0110	Tape search execution Automatic start is not possible as tape search is being executed.	<ul style="list-style-type: none"> • Begin automatic start after the tape search is completed.
0112	Restart pos. return incomplete Automatic start is not possible as the axis has not been returned to the restart position.	<ul style="list-style-type: none"> • Manually return to the restart position. • Turn the automatic restart valid parameter ON, and then execute automatic start.
0113	CNC overheat Automatic start is not possible because a thermal alarm (Z53 CNC overheat) has occurred.	<ul style="list-style-type: none"> • The NC controller temperature has exceeded the specified temperature. • Take appropriate measures to cool the unit.
0115	Cycle st. prohibit(Host comm.) Automatic start cannot be executed as the NC is communicating with the host computer.	<ul style="list-style-type: none"> • Execute automatic start after the communication with the host computer is completed.
0116	Cycle st prohibit(Battery alm) Automatic start cannot be executed because the voltage of the battery inserted in the NC control unit has dropped.	<ul style="list-style-type: none"> • Replace the battery of the NC control unit. • Contact the service center.
0117	R-pnt offset value not set As the reference position offset value has not been set, automatic operation cannot be used.	<ul style="list-style-type: none"> • Perform the initial reference position setting, then set the reference position offset value (#2034 rfpofs).
0138	In absolute position alarm A start signal was input during an absolute position detection alarm.	<ul style="list-style-type: none"> • Reset the absolute position detection alarm, and then input the start signal.
0139	In abs posn initial setting A start signal was input while initializing the absolute position detector's zero point.	<ul style="list-style-type: none"> • Complete zero point initialization before inputting the start signal.
0180	Cycle start prohibit Automatic start is disabled in servo auto turning valid.	<ul style="list-style-type: none"> • Set "0" to "#1164 ATS" when the servo auto turning is not executed.
0190	Cycle start prohibit Automatic start is disabled because setup parameters can be set.	<ul style="list-style-type: none"> • Refer to the manual issued by the machine tool builder.
0191	Cycle start prohibit Automatic start was caused during file deletion or writing.	<ul style="list-style-type: none"> • Cause automatic start after file deletion or writing is completed.
0193	Cycle st. prohibit (Term exp'd) Automatic start is disabled because the valid term has been expired.	<ul style="list-style-type: none"> • Enter the decryption code and turn the power ON again.

(2) Class: T02 Feed hold

The feed hold state been entered due to a condition in the automatic operation.

Error No.	Details	Remedy
0201	H/W stroke end axis exists An axis is at the stroke end.	<ul style="list-style-type: none"> Manually move the axis away from the stroke end limit switch. The machining program must be corrected.
0202	S/W stroke end axis exists An axis is at the stored stroke limit.	<ul style="list-style-type: none"> Manually move the axis. The machining program must be corrected.
0203	Reset signal ON The reset signal has been input.	<ul style="list-style-type: none"> The program execution position has returned to the start of the program. Execute automatic operation from the start of the machining program.
0204	Auto operation pause signal ON The FEED HOLD switch is ON.	<ul style="list-style-type: none"> Resume automatic operation by pressing the "CYCLE START" switch.
0205	Operation mode changed The operation mode changed to another mode during automatic operation.	<ul style="list-style-type: none"> Return to the original automatic operation mode, and resume automatic operation by pressing the "CYCLE START" switch.
0206	Acc/dec time cnst too large The acceleration and deceleration time constants are too large. (This problem occurs at the same time as system alarm Z59.)	<ul style="list-style-type: none"> Increase the set value of the parameter "#1206 G1bF". Decrease the set value of the parameter "#1207 G1btL". Lower the cutting speed.
0215	Abs posn detect alarm occurred An absolute position detection alarm occurred.	<ul style="list-style-type: none"> Reset the absolute position detection alarm.
0220	Aux axis changeover error A movement command was issued to an auxiliary axis.	<ul style="list-style-type: none"> When NC axis control selection signal is ON, automatic operation can be resumed by pressing the "CYCLE START" switch.

(3) Class: T03 Block stop

This indicates that automatic operation stopped after executing one block of the program.

Error No.	Details	Remedy
0301	Single block stop signal ON The SINGLE BLOCK switch on the machine operation panel is ON. The single block or machine lock switch changed.	<ul style="list-style-type: none"> Automatic operation can be resumed by turning the CYCLE START switch ON.
0302	Block stop cmdnd in user macro The block stop command was issued in the user macro program.	<ul style="list-style-type: none"> Automatic operation can be resumed by turning the CYCLE START switch ON.
0303	Operation mode changed The automatic mode changed to another automatic mode.	<ul style="list-style-type: none"> Return to the original automatic operation mode, and resume automatic operation by turning the CYCLE START switch ON.
0304	MDI completed The last block of MDI was completed.	<ul style="list-style-type: none"> Set MDI again, and turn the CYCLE START switch ON to resume MDI operation.
0305	Block start interlock The interlock signal that locks the block start is entered.	<ul style="list-style-type: none"> Check the sequence program.
0306	Cutting blk start interlock The interlock signal that locks the block cutting start is entered.	<ul style="list-style-type: none"> Check the sequence program.
0310	Inclined Z offset change Whether to validate the offset of the inclined Z-axis switched during program operation.	<ul style="list-style-type: none"> Automatic operation can be restarted by turning ON the CYCLE START switch.
0330	Aux axis changeover error NC axis control selection signal was OFF while traveling NC axis.	<ul style="list-style-type: none"> When NC axis control selection signal is ON, automatic operation can be resumed by pressing the "CYCLE START" switch.

(4) Class: T04 Collation stop

This indicates that collation stop was applied during automatic operation.

Error No.	Details	Remedy
0401	Collation stop Collation stop occurred.	<ul style="list-style-type: none"> Automatic operation can be restarted with automatic start.

(5) Class: T10 Fin wait

This indicates the operation state when an alarm did not occur during automatic operation, and nothing seems to have happened.

Error No.	Details																																																																																																																																																							
0□□□	<p>The error number is displayed while each of the completion wait modes listed in the table below is ON. It disappears when the mode is canceled.</p> <div style="text-align: center; margin-bottom: 10px;"> </div> <table border="1" style="display: inline-table; margin-right: 20px;"> <thead> <tr> <th>Alarm No.</th> <th>Unclamp signal wait (Note 2)</th> <th>In dwell execution</th> </tr> </thead> <tbody> <tr><td>0</td><td></td><td></td></tr> <tr><td>1</td><td></td><td>×</td></tr> <tr><td>8</td><td>×</td><td></td></tr> <tr><td>9</td><td>×</td><td>×</td></tr> </tbody> </table> <table border="1" style="display: inline-table;"> <thead> <tr> <th>Alarm No.</th> <th>Door open (Note 1)</th> <th>Waiting for spindle position to be looped</th> <th>Alarm No.</th> <th>Waiting for spindle orientation to complete</th> <th>Waiting for cutting speed deceleration</th> <th>Waiting for rapid traverse deceleration</th> <th>Waiting for MSTB completion</th> </tr> </thead> <tbody> <tr><td>0</td><td></td><td></td><td>0</td><td></td><td></td><td></td><td></td></tr> <tr><td>1</td><td></td><td>×</td><td>1</td><td></td><td></td><td></td><td>×</td></tr> <tr><td>8</td><td>×</td><td></td><td>2</td><td></td><td></td><td>×</td><td></td></tr> <tr><td>9</td><td>×</td><td>×</td><td>3</td><td></td><td></td><td>×</td><td>×</td></tr> <tr><td></td><td></td><td></td><td>4</td><td></td><td>×</td><td></td><td></td></tr> <tr><td></td><td></td><td></td><td>5</td><td></td><td>×</td><td></td><td>×</td></tr> <tr><td></td><td></td><td></td><td>6</td><td></td><td>×</td><td>×</td><td></td></tr> <tr><td></td><td></td><td></td><td>7</td><td></td><td>×</td><td>×</td><td>×</td></tr> <tr><td></td><td></td><td></td><td>8</td><td>×</td><td></td><td></td><td></td></tr> <tr><td></td><td></td><td></td><td>9</td><td>×</td><td></td><td></td><td>×</td></tr> <tr><td></td><td></td><td></td><td>A</td><td>×</td><td></td><td>×</td><td></td></tr> <tr><td></td><td></td><td></td><td>B</td><td>×</td><td></td><td>×</td><td>×</td></tr> <tr><td></td><td></td><td></td><td>C</td><td>×</td><td>×</td><td></td><td></td></tr> <tr><td></td><td></td><td></td><td>D</td><td>×</td><td>×</td><td></td><td>×</td></tr> <tr><td></td><td></td><td></td><td>E</td><td>×</td><td>×</td><td>×</td><td></td></tr> <tr><td></td><td></td><td></td><td>F</td><td>×</td><td>×</td><td>×</td><td>×</td></tr> </tbody> </table> <p>(Note 1) This mode is enabled by the door interlock function. (Note 2) The system is waiting for the index table indexing unclamp signal to turn ON or OFF</p>	Alarm No.	Unclamp signal wait (Note 2)	In dwell execution	0			1		×	8	×		9	×	×	Alarm No.	Door open (Note 1)	Waiting for spindle position to be looped	Alarm No.	Waiting for spindle orientation to complete	Waiting for cutting speed deceleration	Waiting for rapid traverse deceleration	Waiting for MSTB completion	0			0					1		×	1				×	8	×		2			×		9	×	×	3			×	×				4		×						5		×		×				6		×	×					7		×	×	×				8	×							9	×			×				A	×		×					B	×		×	×				C	×	×						D	×	×		×				E	×	×	×					F	×	×	×	×
Alarm No.	Unclamp signal wait (Note 2)	In dwell execution																																																																																																																																																						
0																																																																																																																																																								
1		×																																																																																																																																																						
8	×																																																																																																																																																							
9	×	×																																																																																																																																																						
Alarm No.	Door open (Note 1)	Waiting for spindle position to be looped	Alarm No.	Waiting for spindle orientation to complete	Waiting for cutting speed deceleration	Waiting for rapid traverse deceleration	Waiting for MSTB completion																																																																																																																																																	
0			0																																																																																																																																																					
1		×	1				×																																																																																																																																																	
8	×		2			×																																																																																																																																																		
9	×	×	3			×	×																																																																																																																																																	
			4		×																																																																																																																																																			
			5		×		×																																																																																																																																																	
			6		×	×																																																																																																																																																		
			7		×	×	×																																																																																																																																																	
			8	×																																																																																																																																																				
			9	×			×																																																																																																																																																	
			A	×		×																																																																																																																																																		
			B	×		×	×																																																																																																																																																	
			C	×	×																																																																																																																																																			
			D	×	×		×																																																																																																																																																	
			E	×	×	×																																																																																																																																																		
			F	×	×	×	×																																																																																																																																																	

6.1.3 Servo/Spindle Alarms

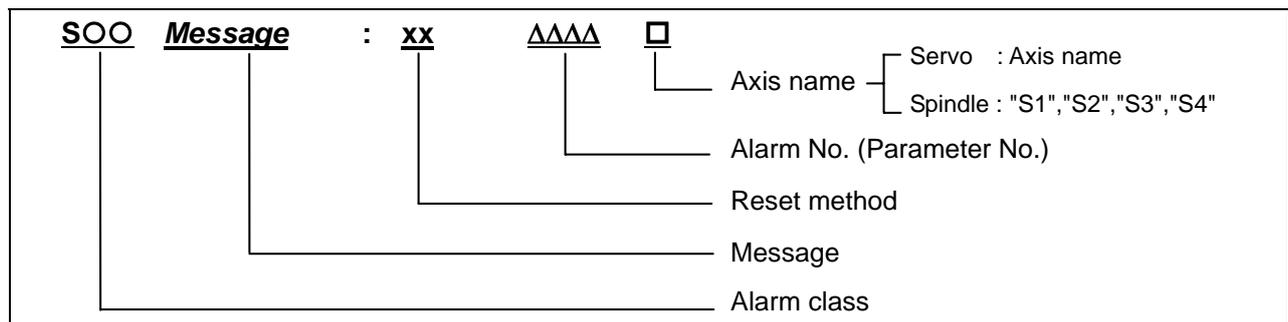
This section describes alarms occurred by the errors in the servo system such as the drive unit, motor and encoder, etc. The alarm message, alarm No. and axis name will display on the alarm message screen. The axis where the alarm occurred and the alarm No. will also display on the servo monitor screen and the spindle monitor screen respectively. If several alarms have occurred, up to two errors per axis will display on the servo monitor screen and the spindle monitor screen respectively.

(Note 1) The alarm class and alarm reset method combinations are preset.

Alarm class	Reset method	Resetting methods
S01	PR	After removing the cause of the alarm, reset the alarm by turning the NC power ON again.
S02	PR	After correcting the parameter, reset the alarm by turning ON the NC power again.
S03	NR	After removing the cause of the alarm, reset the alarm by inputting the NC RESET key.
S04	AR	After removing the cause of the alarm, reset the alarm by turning the drive unit power ON again.
S51	-	This is cleared if a correct value is set.
S52	-	-

(Note 2) The resetting method may change according to the alarm class.

For example, even if "S03 SERVO ALARM: NR" is displayed, it may be necessary to turn the NC power ON again.



(1) Class: S01/S03/S04 Servo alarm

(a) Servo drive unit alarm

No.	Message	Details	Reset method	Stop method
0010	Insufficient voltage	A drop of bus voltage was detected in main circuit.	PR	Dynamic stop
0011	Axis selection error	The axis selection rotary switch has been incorrectly set.	AR	Initial error
0012	Memory error 1	A hardware error was detected during the power ON self-check.	AR	Initial error
0013	Software processing error 1	An error was detected for the software execution state.	PR	Dynamic stop
0016	Init mag pole pos detect err	The initial magnetic pole position, detected in the initial magnetic pole position detection control, is not reliable. In the DC excitation function, this error will be detected when the servo ON has been set before the magnetic pole shift amount is set while the absolute position detector is used.	PR	Dynamic stop
0017	A/D converter error	A current feedback error was detected.	PR	Dynamic stop
0018	Motor side dtc: Init commu err	An error was detected in the initial communication with the motor side detector.	PR	Initial error
001A	Machine side dtc: Init comu er	An error was detected in the initial communication with the machine side detector.	PR	Initial error
001B	Machine side dtc: Error 1	An error was detected by the detector connected to the machine side. The error details are different according to the connected detector. Refer to "Detector alarm".		Dynamic stop
001C	Machine side dtc: Error 2			
001D	Machine side dtc: Error 3			
001E	Machine side dtc: Error 4			
001F	Machine side dtc: Commu error			
0021	Machine side dtc: No signal	An error was detected in the ABZ-phase in the full closed loop control system.	PR	Dynamic stop
0024	Grounding	The motor power cable is in contact with FG (Frame Ground).	PR	Dynamic stop
0025	Absolute position data lost	The absolute position was lost in the detector.	AR	Initial error
0026	Unused axis error	A power module error was detected on the axis set to Free.	PR	Dynamic stop
0027	Machine side dtc: Error 5	An error was detected by the detector connected to the machine side. The error details are different according to the connected detector. Refer to "Detector alarm".		Dynamic stop
0028	Machine side dtc: Error 6			
0029	Machine side dtc: Error 7			
002A	Machine side dtc: Error 8			
002B	Motor side dtc: Error 1			
002C	Motor side dtc: Error 2			
002D	Motor side dtc: Error 3			
002E	Motor side dtc: Error 4			
002F	Motor side dtc: Commu error	An error was detected in the communication with the motor side detector.		
0030	Over regeneration	Over-regeneration level exceeded 100%. The regenerative resistor is overloaded.	PR	Dynamic stop
0031	Overspeed	The motor speed exceeded the allowable speed.	PR	Deceleration stop enabled
0032	Power module overcurrent	The power module detected the overcurrent.	PR	Dynamic stop
0033	Overvoltage	The bus voltage in main circuit exceeded the allowable value.	PR	Dynamic stop
0034	NC-DRV commu: CRC error	An error was detected in the data received from the NC.	PR	Deceleration stop enabled

No.	Message	Details	Reset method	Stop method
0035	NC command error	The travel command data received from the NC was excessive.	PR	Deceleration stop enabled
0036	NC-DRV commu: Commu error	The communication with the NC was interrupted.	PR	Deceleration stop enabled
0037	Initial parameter error	An incorrect set value was detected among the parameters send from the NC at the power ON.	PR	Initial error
0038	NC-DRV commu: Protocol error 1	An error was detected in the communication frames received from the NC.	PR	Deceleration stop enabled
0039	NC-DRV commu: Protocol error 2	An error was detected in the axis data received from the NC.	PR	Deceleration stop enabled
003A	Overcurrent	Excessive motor drive current was detected.	PR	Dynamic stop
003B	Power module overheat	The power module detected an overheat.	PR	Dynamic stop
003C	Regeneration circuit error	An error was detected in the regenerative transistor or in the regenerative resistor.	PR	Dynamic stop
003D	Pw sply volt err acc/dec	A motor control error, due to an input voltage failure, was detected.	PR	Dynamic stop
003E	Magnet pole pos detect err	The magnetic pole position, detected in the magnetic pole position detection control, is not reliable.	AR	Dynamic stop
0041	Feedback error 3	Either a missed feedback pulse in the motor side detector or an error in the Z-phase was detected in the full closed loop system.	PR	Dynamic stop
0042	Feedback error 1	Either a missed feedback pulse in the detector used for the position detection or an error in the Z-phase was detected.	PR	Dynamic stop
0043	Feedback error 2	An excessive difference in feedback was detected between the machine side detector and the motor side detector.	PR	Dynamic stop
0045	Fan stop	An overheat of the power module was detected during the cooling fan stopping.	PR	Dynamic stop
0046	Motor overheat	Either the motor or the motor side detector detected an overheat.	NR	Deceleration stop enabled
0048	Motor side dtc: Error 5	An error was detected by the detector connected to the motor side. The error details are different according to the connected detector. Refer to "Detector alarm".		Dynamic stop
0049	Motor side dtc: Error 6			
004A	Motor side dtc: Error 7			
004B	Motor side dtc: Error 8			
004F	Instantaneous power interrupt	The control power supply has been shut down for 50ms or more.	NR	Deceleration stop enabled
0050	Overload 1	Excessive load current was detected.	NR	Deceleration stop enabled
0051	Overload 2	Excessive load current was detected.	NR	Deceleration stop enabled
0052	Excessive error 1	A position tracking error was detected. (during servo ON)	NR	Deceleration stop enabled
0053	Excessive error 2	A position tracking error was detected. (during servo OFF)	NR	Dynamic stop
0054	Excessive error 3	The anomalous motor current was detected at the detection of Excessive error 1.	NR	Dynamic stop
0058	Collision detection 1: G0	A disturbance torque exceeded the tolerable disturbance torque in rapid traverse modal (G0). The tolerable disturbance torque is decided by SV060:TLMT.	NR	Maximum capacity deceleration stop
0059	Collision detection 1: G1	A disturbance torque exceeded the tolerable disturbance torque in the cutting feed modal (G1). The tolerable disturbance torque is decided by SV060:TLMT and SV035:SSF4/clG1(bitC, bitD and bitE).	NR	Maximum capacity deceleration stop
005A	Collision detection 2	A current command with the maximum capacity current value was detected.	NR	Maximum capacity deceleration stop
005B	Sfty obsrvation: Cmd spd err	A commanded speed exceeding the safe speed was detected in speed monitoring mode.	PR	Deceleration stop enabled

No.	Message	Details	Reset method	Stop method
005D	Sfty obsrvation: Door stat err	The door state signal input in the NC does not coincide with the door state signal input in the drive unit. Otherwise, door open state was detected in normal mode.	PR	Deceleration stop enabled
005E	Sfty obsrvation: FB speed err	A motor speed exceeding the safe speed was detected in the speed monitoring mode.	PR	Deceleration stop enabled
005F	External contactor error	A contact of the external contactor is welding.	NR	Deceleration stop enabled
0060 ~ 0077	Power supply alarm	The power supply unit detected an error. The error details are different according to the connected power supply unit. Refer to "Power supply alarm".		Dynamic stop
0080	Motor side dtc: cable err	A difference of type was detected between the motor side detector and the cable connected to the detector. Otherwise, the cable type for the motor side detector was not successfully achieved.	AR	Initial error
0081	Machine side dtc: cable err	A difference of type was detected between the machine side detector and the cable connected to the detector. Otherwise, the cable type for the machine side detector was not successfully achieved.	AR	Initial error
0087	Drive unit communication error	The communication frame between drivers was aborted.	PR	Dynamic stop
0088	Watchdog	The drive unit does not operate correctly.	AR	Dynamic stop
008A	Drivers commu data error 1	The communication data 1 between drivers exceeded the tolerable value in the communication between drivers.	PR	Dynamic stop
008B	Drivers commu data error 2	The communication data 2 between drivers exceeded the tolerable value in the communication between drivers.	PR	Dynamic stop

(Note1) Definitions of terms in the table are as follows.

Motor side detector: Detector connected to CN2 Machine side detector: Detector connected to CN3

(Note2) Resetting methods

NR: Reset with the NC RESET button. This alarm can also be reset with the PR and AR resetting conditions.

PR: Reset by turning the NC power ON again. This alarm can also be reset with the AR resetting conditions.

When the control axis is removed, this alarm can be reset with the NC RESET button. (Excluding alarms 32 and 37.)

AR: Reset by turning the servo drive unit power ON again.

Separate table : Detector alarm (Servo drive unit)

Alarm number when the detector is connected to the motor side	2B	2C	2D	2E	48	49	4A	4B	
Alarm number when the detector is connected to the machine side	1B	1C	1D	1E	27	28	29	2A	
OSA17 OSE104, OSE105 OSA104, OSA105 OSA405, OSA166	MITSUBISHI	Memory alarm	LED alarm	Data alarm	-	-	-	-	
OSA18		CPU alarm	-	Data alarm	-	-	-	-	
MDS-B-HR		Memory error	-	Data error	-	Scale not connected	-	-	
AT343, AT543	Mitutoyo	Initialization error	EEPROM error	Photoelectric type, static capacity type data mismatch	ROM/RAM error	CPU error	Photoelectric type overspeed	Static capacity type error	Photoelectric type error
LC191M, LC491M RCN723, RCN223 APE391M	HEIDENHAIN	Initialization error	EEPROM error	Relative/absolute position data mismatch	ROM/RAM error	CPU error	Overspeed	Absolute position data error	Relative position data error
Futaba absolute position scale	Futaba	-	-	-	-	-	-	Waveform error	Overspeed Absolute position is lost
MP scale, MPI scale	Mitsubishi Heavy Industries	Installation accuracy fault	-	Detection position deviance	Scale breaking	Absolute value detection fault	-	Gain fault	Phase fault
MJ831	SONY	-	-	-	-	-	-	-	Detector alarm

(Note1) Definitions of terms in the table are as follows.

Motor side detector: Detector connected to CN2 Machine side detector: Detector connected to CN3

(Note2) A driver processes all reset types of alarms as "PR". However, "AR" will be applied according to the detector.

(b) Spindle drive unit alarm

No.	Message	Details	Reset method	Stop method
0010	Insufficient voltage	A drop of bus voltage was detected in main circuit.	PR	Coast to a stop
0011	Axis selection error	The axis selection rotary switch has been incorrectly set.	AR	Initial error
0012	Memory error 1	A hardware error was detected during the power ON self-check.	AR	Initial error
0013	Software processing error 1	An error was detected for the software execution state.	PR	Coast to a stop
0016	Init mag pole pos detect err	The magnetic pole position, detected in the initial magnetic pole position detection control, is not reliable.	PR	Coast to a stop
0017	A/D converter error	A current feedback error was detected.	PR	Coast to a stop
0018	Motor side dtc: Init commu err	An error was detected in the initial communication with the motor side detector.	PR	Initial error
0019	Detector commu err in syn cont	An error was detected in the communication with the extended connection detector.	PR	Coast to a stop
001A	Machine side dtc: Init comu er	An error was detected in the initial communication with the machine side detector.	PR	Initial error
001B	Machine side dtc: Error 1	An error was detected by the detector connected to the machine side. The error details are different according to the connected detector. Refer to "Detector alarm".		Coast to a stop
001C	Machine side dtc: Error 2			
001D	Machine side dtc: Error 3			
001E	Machine side dtc: Error 4			
001F	Machine side dtc: Commu error	An error was detected in the communication with the machine side detector.	PR	Coast to a stop
0020	Motor side dtc: No signal	The cable type of the motor side detector does not coincide with the detector type set with the parameter.	PR	Initial error
0021	Machine side dtc: No signal Excessive speed error	The cable type of the machine side detector does not coincide with the detector type set with the parameter.	PR	Initial error
		An error was detected in the ABZ-phase in the full closed loop control system.	PR	Coast to a stop
0023	Grounding	An excessive speed tracking error was detected (during servo ON).	NR	Coast to a stop
0024	Machine side dtc: No signal	A grounding of the motor power cable or motor was detected.	AR	Coast to a stop
0026	Unused axis error	A power module error was detected on the axis set to Free.	PR	Coast to a stop
0027	Machine side dtc: Error 5	An error was detected by the detector connected to the machine side. The error details are different according to the connected detector. Refer to "Detector alarm".		Coast to a stop
0028	Machine side dtc: Error 6			
0029	Machine side dtc: Error 7	An error was detected by the detector connected to the motor side. The error details are different according to the connected detector. Refer to "Detector alarm".		Coast to a stop
002A	Machine side dtc: Error 8			
002B	Motor side dtc: Error 1			
002C	Motor side dtc: Error 2			
002D	Motor side dtc: Error 3	An error was detected in the communication with the motor side detector.	PR	Coast to a stop
002E	Motor side dtc: Error 4			
002F	Motor side dtc: Commu error			
0030	Over regeneration	Over-regeneration level exceeded 100%. The regenerative resistor is overloaded.	PR	Coast to a stop
0031	Overspeed	The motor speed exceeded the allowable speed.	PR	Deceleration stop enabled
0032	Power module overcurrent	The power module detected the overcurrent.	PR	Coast to a stop
0033	Overvoltage	The bus voltage in main circuit exceeded the allowable value.	PR	Coast to a stop
0034	NC-DRV commu: CRC error	An error was detected in the data received from the NC.	PR	Deceleration stop enabled
0035	NC command error	An error was detected in the travel command data received from the NC.	PR	Deceleration stop enabled
0036	NC-DRV commu: Commu error	The communication with the NC was interrupted.	PR	Deceleration stop enabled
0037	Initial parameter error	An incorrect set value was detected among the parameters send from the NC at the power ON.	PR	Initial error

No.	Message	Details	Reset method	Stop method
0038	NC-DRV commu: Protocol error 1	An error was detected in the communication frames received from the NC.	PR	Deceleration stop enabled
0039	NC-DRV commu: Protocol error 2	An error was detected in the axis data received from the NC.	PR	Deceleration stop enabled
003A	Overcurrent	Excessive motor drive current was detected.	PR	Coast to a stop
003B	Power module overheat	The power module detected an overheat.	PR	Coast to a stop
003C	Regeneration circuit error	An error was detected in the regenerative transistor or in the regenerative resistor.	PR	Coast to a stop
003E	Magnet pole pos detect err	The magnetic pole position, detected in the magnetic pole position detection control, is not reliable.	AR	Coast to a stop
0041	Feedback error 3	An error was detected in the feedback of the motor side detector.	PR	Coast to a stop
0042	Feedback error 1	An error was detected in the feedback of the machine side detector.	PR	Coast to a stop
0043	Feedback error 2	An excessive difference in feedback was detected between the motor side detector and the machine side detector.	PR	Coast to a stop
0045	Fan stop	A cooling fan in the drive unit stopped.	PR	Coast to a stop
0046	Motor overheat	Either the motor or the motor side detector detected an overheat.	NR	Deceleration stop enabled
0048	Motor side dtc: Error 5	An error was detected by the detector connected to the motor side.		Coast to a stop
0049	Motor side dtc: Error 6	The error details are different according to the connected detector.		
004A	Motor side dtc: Error 7	Refer to "Detector alarm".		
004B	Motor side dtc: Error 8			
004C	Current err mag pole estim	Current detection failed at the pulse-applied magnetic pole estimation by IPM spindle motor.	NR	Coast to a stop
004E	NC command mode error	An error was detected in the spindle control mode send from the NC.	NR	Deceleration stop enabled
004F	Instantaneous power interrupt	The control power supply has been shut down for 50ms or more.	NR	Deceleration stop enabled
0050	Overload 1	Excessive load current was detected.	NR	Deceleration stop enabled
0051	Overload 2	Excessive load current was detected.	NR	Deceleration stop enabled
0052	Excessive error 1	A position tracking error was detected. (during servo ON)	NR	Deceleration stop enabled
0054	Excessive error 3	The anomalous motor current was detected at the detection of Excessive error 1.	NR	Coast to a stop
005B	Sfty obsrvation: Cmd spd err	A commanded speed exceeding the safe speed was detected in speed monitoring mode.	PR	Deceleration stop enabled
005D	Sfty obsrvation: Door stat err	The door state signal input in the NC does not coincide with the door state signal input in the drive unit. Otherwise, door open state was detected in normal mode.	PR	Deceleration stop enabled
005E	Sfty obsrvation: FB speed err	A motor speed exceeding the safe speed was detected in the speed monitoring mode.	PR	Deceleration stop enabled
005F	External contactor error	A contact of the external contactor is welding.	NR	Deceleration stop enabled
0060 ~ 0077	Power supply alarm	The power supply unit detected an error. The error details are different according to the connected power supply unit. Refer to "Power supply alarm".		Coast to a stop
0080	Motor side dtc: cable err	The connected cable type does not coincide with the motor side detector type.	PR	Initial error
0081	Machine side dtc: cable err	The connected cable type does not coincide with the machine side detector type.	PR	Initial error
0087	Drive unit communication error	The communication frame between drivers was aborted.	PR	Coast to a stop
0088	Watchdog	The drive unit does not operate correctly.	AR	Coast to a stop
008A	Drivers commu data error 1	The communication data 1 between drivers exceeded the tolerable value in the communication between drivers.	PR	Coast to a stop
008B	Drivers commu data error 2	The communication data 2 between drivers exceeded the tolerable value in the communication between drivers.	PR	Coast to a stop

(Note)

Resetting methods

NR: Reset with the NC RESET button. This alarm can also be reset with the PR and AR resetting conditions.

PR: Reset by turning the NC power ON again. This alarm can also be reset with the AR resetting conditions.

When the control axis is removed, this alarm can be reset with the NC RESET button. (Excluding alarms 32 and 37.)

AR: Reset by turning the servo drive unit power ON again.

Separate table : Detector alarm (Spindle drive unit)

Alarm number when the detector is connected to the motor side		2B	2C	2D	2E	48	49	4A	4B
Alarm number when the detector is connected to the machine side		1B	1C	1D	1E	27	28	29	2A
TS5690 TS5691	MITSUBISHI	Memory error	Waveform error	-	-	-	Overspeed	-	Relative position data error
MDS-B-HR		Initialization error	-	Data error	-	Connection error	-	-	-
OSA18		CPU error	-	Data error	-	-	-	-	-
ERM280 + APE391M	HEIDENHAIN	Initialization error	EEPROM error	-	-	CPU error	Overspeed	-	Relative position data error
MPCI scale	Mitsubishi Heavy Industries	Installation accuracy fault	-	Detection position deviance	Scale breaking	-	-	Gain fault	Phase fault

(Note) A driver processes all reset types of alarms as "PR". However, "AR" will be applied according to the detector.

(c) Power supply alarm

No.	LED display	Message	Details	Reset method
0061		Pw sply: Pwr module overcurnt	Overcurrent protection function in the power module has started its operation.	PR
0062		Pw sply: Frequency error	The input power supply frequency increased above the specification range.	PR
0067		Pw sply: Phase interruption	An open-phase condition was detected in input power supply circuit.	PR
0068		Pw sply: Watchdog	The system does not operate correctly.	AR
0069		Pw sply: Grounding	The motor power cable is in contact with FG (Frame Ground).	PR
006A		Pw sply: Ext contactor weld	A contact of the external contactor is welding.	PR
006B		Pw sply: Rush relay welding	A resistor relay for rush short circuit fails to be OFF.	PR
006C		Pw sply: Main circuit error	An error was detected in charging operation of the main circuit capacitor.	PR
006E		Pw sply: Memory error/AD error	An error was detected in the internal memory or A/D converter.	AR
006F		Power supply error	No power supply is connected to the drive unit, or a communication error was detected.	AR
0070		Pw sply: Ext emergency stp err	A mismatch of the external emergency stop input and NC emergency stop input continued for 30 seconds.	PR
0071		Pw sply: Instant pwr interrupt	The power was momentarily interrupted.	NR
0072		Pw sply: Fan stop	A cooling fan built in the power supply unit stopped, and overheat occurred in the power module.	PR
0073		Pw sply: Over regeneration	Over-regeneration detection level became over 100%. The regenerative resistor is overloaded. This alarm cannot be reset for 15 min from the occurrence to protect the regeneration resistor. Leave the drive system energized for more than 15 min, then turn the power ON to reset the alarm.	NR
0075		Pw sply: Overvoltage	L+ and L- bus voltage in main circuit exceeded the allowable value. As the voltage between L+ and L- is high immediately after this alarm, another alarm may occur if this alarm is reset in a short time. Wait more than 5 min before resetting so that the voltage drops.	NR
0076		Pw sply: Ext EMG stop set err	The rotary switch setting of external emergency stop is not correct, or a wrong external emergency stop signal is input.	AR
0077		Pw sply: Power module overheat	Thermal protection function in the power module has started its operation.	PR

(Note 1) If a power supply alarm (60 to 77) occurs, all servos will stop with the dynamic brakes, and all spindles will coast to a stop.

(Note 2) "b", "C" and "d" displayed on the power supply unit's LED as a solid light (not flickering) do not indicate an alarm.

(2) Class: S02 Message: Initial parameter error

An error was found in the parameters transmitted from the controller to the drive unit when the power was turned ON.

Remove the cause of the alarm, and then reset the alarm by turning the controller power OFF once.

Alarm No.	Details	Remedy
2201 to 2264	The servo parameter setting data is illegal. The alarm No. is the No. of the servo parameter where the error occurred.	Check the descriptions for the appropriate servo parameters and correct them.
2301	The number of constants to be used in the following functions is too large: <ul style="list-style-type: none"> • Electronic gears • Position loop gain • Speed feedback conversion 	Check that all the related parameters are specified correctly. sv001:PC1, sv002:PC2, sv003:PGN1 sv018:PIT, sv019:RNG1, sv020:RNG2
2302	When high-speed serial incremental detector (OSE104, OSE105) is connected, parameters for absolute position are set to ON. Set the parameters for absolute position detection to OFF. To detect an absolute position, replace the incremental specification detector with an absolute position detector.	Check that all the related parameters are specified correctly. sv017:SPEC, sv025:MTYP
2303	No servo option is found. The closed loop (including the ball screw-end detector) or dual feedback control is an optional function.	Check that all the related parameters are specified correctly. sv025:MTYP/pen sv017:SPEC/dfbx
2304	No servo option is found. The SHG control is an optional function.	Check that all the related parameters are specified correctly. sv057:SHGC sv058:SHGCsp
2305	No servo option is found. The adaptive filtering is an optional function.	Check that all the related parameters are specified correctly. sv027:SSF1/aft
13001 to 13256	Parameter error The spindle parameter setting data is illegal. The alarm No. is the No. of the spindle parameter where the error occurred.	Check the descriptions for the appropriate spindle parameters and correct them. Refer to Alarm No.37 in Spindle Drive Maintenance Manual.

(3) Class: S51 Message: Parameter error

This warning is displayed if a parameter outside the tolerance range is set.

Illegal settings will be ignored.

This alarm will be reset when a correct value is set.

Alarm No.	Details	Remedy
2201 to 2264	Servo parameter setting data is illegal. The alarm No. is the No. of the servo parameter where the warning occurred.	Check the descriptions for the appropriate servo parameters and correct them.
13001 to 13256	Spindle parameter setting data is illegal. The alarm No. is the No. of the spindle parameter where the warning occurred.	Check the descriptions for the appropriate spindle parameters and correct them. Refer to Spindle Drive Maintenance Manual.

(4) Class: S52 Servo warning

When a warning occurs, a warning No. will appear on the NC monitor screen and with the LEDs on the front of the drive unit. Check the warning No., and remove the cause of the warning by following this list.

(a) Servo drive unit warning

No.	Message	Details	Reset method	Stop method
0096	Scale feedback error	An excessive difference in feedback amount was detected between the motor side detector and the MPI scale in MPI scale absolute position detection system.	*	-
0097	Scale offset error	An error was detected in the offset data that is read at the NC power-ON in MPI scale absolute position detection system.	PR	-
009E	Absolute position detector: Revolution counter error	An error was detected in the revolution counter data of the absolute position detector. The accuracy of absolute position is not guaranteed.	*	-
009F	Battery voltage drop	The battery voltage to be supplied to the absolute position detector is dropping.	*	-
00A6	Fan stop warning	A cooling fan in the drive unit stopped.	*	-
00E0	Overregeneration warning	Over-regeneration detection level exceeded 80%.	*	-
00E1	Overload warning	A level of 80% of the Overload 1 alarm state was detected.	*	-
00E4	Set parameter warning	An incorrect set value was detected among the parameters send from the NC in the normal operation.	*	-
00E6	Control axis detachment warning	A control axis is being detached. (State display)	*	-
00E7	In NC emergency stop state	In NC emergency stop. (State display)	*	Deceleration stop enabled
00E8 ~ 00EF	Power supply warning	The power supply unit detected a warning. The error details are different according to the connected power supply unit. Refer to "Power supply warning".	*	- *EA : Deceleration stop enabled

(Note 1) Definitions of terms in the table are as follows.

Motor side detector: Detector connected to CN2 Machine side detector: Detector connected to CN3

(Note 2) Resetting methods

*: Automatically reset once the cause of the warning is removed.

NR: Reset with the NC RESET button. This warning can also be reset with the PR and AR resetting conditions.

PR: Reset by turning the NC power ON again. This warning can also be reset with the AR resetting conditions.

When the control axis is removed, this warning can be reset with the NC RESET button. (Excluding warning 93.)

AR: Reset by turning the servo drive unit power ON again.

(Note 3) Servo and spindle motor do not stop when the warning occurs.

(Note 4) When an emergency stop is input, servo and spindle motor decelerate to a stop.

(When SV048, SV055 or SV056 is set for servo and when SP055 or SP056 is set for spindle.)

(b) Spindle drive unit warning

No.	Message	Details	Reset method	Stop method
00A6	Fan stop warning	A cooling fan in the drive unit stopped.	*	-
00E0	Overregeneration warning	Over-regeneration detection level exceeded 80%.	*	-
00E1	Overload warning	A level of 80% of the Overload 1 alarm state was detected.	*	-
00E4	Set parameter warning	A parameter was set to the value over the setting range.	*	-
00E6	Control axis detachment warning	A control axis is being detached. (State display)	*	-
00E7	In NC emergency stop state	In NC emergency stop. (State display)	*	Deceleration stop enabled
00E8 ~ 00EF	Power supply warning	The power supply unit detected a warning. The error details are different according to the connected power supply unit. Refer to "Power supply warning".	*	-

(Note 1) Resetting methods

*: Automatically reset once the cause of the warning is removed.

NR: Reset with the NC RESET button. This warning can also be reset with the PR and AR resetting conditions.

PR: Reset by turning the NC power ON again. This warning can also be reset with the AR resetting conditions.

When the control axis is removed, this warning can be reset with the NC RESET button. (Excluding warning 93.)

AR: Reset by turning the servo drive unit power ON again.

(Note 2) Servo and spindle motor do not stop when the warning occurs.

(Note 3) When an emergency stop is input, servo and spindle motor decelerate to a stop.

(When SV048, SV055 or SV056 is set for servo and when SP055 or SP056 is set for spindle.)

(c) Power supply warning

No.	LED display	Message	Details	Reset method
00E9		Instantaneous power interruption warning	The power was momentarily interrupted.	NR
00EA		In external emergency stop state	External emergency stop signal was input.	*
00EB		Power supply: Over regeneration warning	Over-regeneration detection level exceeded 80%.	*
00EE		Pw sply: Fan stop warning	A cooling fan built in the power supply unit stopped.	*

(Note 1) Resetting methods

*: Automatically reset once the cause of the warning is removed.

NR: Reset with the NC RESET button. This warning can also be reset with the PR and AR resetting conditions.

PR: Reset by turning the NC power ON again. This warning can also be reset with the AR resetting conditions.

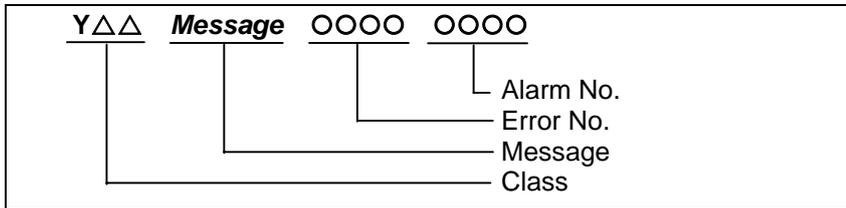
When the control axis is removed, this warning can be reset with the NC RESET button. (Excluding warning 93.)

AR: Reset by turning the servo drive unit power ON again.

(Note 2) Servo and spindle motor do not stop when the warning occurs.

6.1.4 MCP Alarm

An error has occurred in the drive unit and other interfaces. (The bold characters are the messages displayed on the screen.)



(1) Class: Y02 System alarm

An error occurred in the data transmitted between the MCP and drive unit after the power was turned ON.

Error No.	Details		Remedy
0050	System alm: Process time over		The software or hardware may be damaged. Contact the service center.
0051	Alarm No.		A communication error has occurred between the controller and drive unit. <ul style="list-style-type: none"> • Take measures against noise. • Check that the communication cable connector between the controller and drive unit and one between the drive units are tight. • Check whether the communication cable between the controller and drive unit and one between the drive units are disconnected. • A drive unit may be faulty. Take a note of the 7-segment LED contents of each drive unit and report to the Service Center. • Update the drive unit software version.
	0000	SV commu er: CRC error 1 (10 times/910.2 ms)	
	0001	SV commu er: CRC error 2 (2 continuous times)	
	0002	SV commu er: Recv timing err (2 continuous times)	
	xy03	SV commu er: Data ID error (2 continuous times) x: Channel No. (0 to) y: Drive unit rotary switch No. (0 to)	
	xy04	SV commu er: Recv frame No. (2 continuous times) x: Channel No. (0 to) y: Number of reception frame -1 (0 to)	
	x005	SV commu er: Commu error (No error classification) x: Channel No. (0 to)	
	x006	SV commu er: Connect error x: Channel No. (0 to)	
	xy20	SV commu er: Init commu error The drive unit could not shift to the initial communication run time and stopped. x: Channel No. (0 to) y: Drive unit rotary switch No. (0 to)	
	xy30	SV commu er: Node detect error No response from drive unit to the request from NC when setting network configuration. x: Channel No. (0 to) y: Station No. (0 to)	
xy31	SV commu er: Commu not support Drive unit's software version doesn't support the communication mode that the controller requires. x: Channel No. (0 to) y: Station No. (0 to)		

(Note) When two or more "Y02 System alarms" occur at the same time, only the alarm which occurs first is displayed.

(2) Class: Y03 Message: Drive unit unequipped

The drive unit is not correctly connected.

Error No.	Details	Remedy
Alphabet (axis name)	Servo axis drive unit not mounted	Check the drive unit mounting state. <ul style="list-style-type: none"> • Check the end of the cable wiring. • Check the cable for broken wires. • Check the connector insertion. • The drive unit input power is not being input. • The drive unit axis No. switch is illegal.
1 to 4	PLC axis drive unit not mounted	
S	No.1 spindle axis drive unit not mounted	
T	No.2 spindle axis drive unit not mounted	
M	No.3 spindle axis drive unit not mounted	
N	No.4 spindle axis drive unit not mounted	

(3) Class : Y05 Message: Initial parameter error

Details	Remedy
There is a problem in the value set for the number of axes or the number of part systems.	Check the value set for the corresponding parameters. #1001 SYS_ON #1002 axisno #1039 spinno, etc.

(4) Class: Y06 Message: mcp_no setting error

There are differences in the MCP and axis parameters when the NC power is turned ON.

Error No.	Details	Remedy
0001	There is a skipped number in the channels.	Check the values set for the following parameters. #1021 mcp_no #3031 smcp_no
0002	The random layout setting is duplicated.	
0003	The drive unit fixed setting "0000" and random layout setting "****" are both set.	
0004	The spindle/C axis "#1021 mcp_no" and "#3031 smcp_no" are not set to the same values.	
0005	A random layout is set for the "#1154 pdoor" =1 two-part system.	
0006	The channel No. parameter is not within the setting range.	

(5) Class: Y07 Message: Too many axes connected

The number of connected axes exceeds the number allowed in the system.

(Alarm No.) 		
Alarm No.	Details	Remedy
0000 to 00FF	The number of axes connected to each channel exceeds the maximum number of connectable axes. The exceeded number of axes per channel is displayed as alarm No. This alarm occurs when the drive unit is connected only with the 2nd channel without connecting with the 1st channel.	Remove connected axes from the channel whose alarm No. is other than 0 for the number displayed as the alarm No. Keep the number of connected axes to or less than the maximum that can be connected.

(Note 1) The number of axes is limited per each drive unit interface channel.

(Note 2) Maximum number of axes that can be connected differs depending on whether or not an expansion unit is available or the setting of "#11012 16 axes for 1ch". The maximum number of connectable axes is as shown below.

Extension unit	#11012 16 axes for 1ch	Maximum number of axes to be connected (Per 1 channel)
Yes	0/1	8 axes
No	0	
		1

(Note 3) If this alarm occurs, the alarm "Y03 Message: Drive unit unequipped" will not occur.

(Note 4) This alarm is displayed taking precedence over the alarm "Y08 Too many drive units connected" and "Y09 Too many axisno connected".

(6) Class: Y08 Message: Too many drive units connected

The number of connected drive units exceeds the number allowed in the system.

(Alarm No.) 		
Alarm No.	Details	Remedy
0000 to 00FF	The number of drive units connected to each channel exceeds 8. The exceeded number of drive units per channel is displayed as alarm No.	Remove drive units from the channel whose alarm No. is other than 0 for the number displayed as the alarm No. Keep the number of connected drive units to 8 or less.

(Note 1) The drive unit is not counted when all the axes connected to it are invalid.

(Note 2) If this alarm occurs, the alarm "Y03 Message: Drive unit unequipped" will not occur.

(Note 3) The alarm "Y07 Too many axes connected" and "Y09 Too many axisno connected" are displayed taking precedence over this alarm.

(7) Class: Y09 Message: Too many axisno connected

The connected axes No. (drive unit's rotary switch No.) is bigger than the No. allowed in the system.

(Alarm No.) 		
Alarm No.	Details	Remedy
0000 to 0011	The No. of the axis (drive unit's rotary switch No.) connected to each channel is bigger than the No. allowed.	For the channel whose alarm No. is 1, keep the axis No. (drive unit's rotary switch No.) not bigger than the No. allowed.

(Note 1) The axis No. is limited per each drive unit interface channel.

(Note 2) The biggest allowed connected axis No. differs depending on whether or not an expansion unit is available or the setting of "#11012 16 axes for 1ch". The biggest connectable axis No. is as shown below.

Extension unit	#11012 16 axes for 1ch	Highest allowed connected axis No. (Per 1 channel)
Yes	0/1	0 to 7
No	0	
	1	0 to F

(Note 3) If this alarm occurs, the alarm "Y03 Message: Drive unit unequipped" will not occur.

(Note 4) This alarm is displayed taking precedence over the alarm "Y08 Too many drive units connected".

(Note 5) The alarm "Y07 Too many axes connected" is displayed taking precedence over this alarm.

(8) Class: Y12 Message: No commu. with axis drv unit

Details	Remedy
Even though the high-speed synchronous tapping option is valid, drive unit that doesn't support the option is connected.	Replace it with a drive unit that supports the option.

(9) Class: Y13 Message: No commu. with sp drv unit

Details	Remedy
Even though the high-speed synchronous tapping option is valid, drive unit that doesn't support the option is connected.	Replace it with a drive unit that supports the option.

(10) Class: Y14 Message: Comm btwn drives not ready

Details	Remedy
Communication of drive units failed to be ready within a specified time.	<ul style="list-style-type: none"> • Connection of drive units may be wrong. • Check if any of drive units is broken.

(11) Class: Y20 Safety observation alarm

When this alarm is output, emergency stop mode is applied.

Refer to "remedy" of each alarm as to how to cancel the alarm.

Error No.	Alarm No.	Details	Remedy
0001	Axis name	Parameter compare error The speed monitoring parameter in the NC and the parameter transmitted to the drive unit are not matched. The name of the axis with an error is displayed.	The NC or the servo drive unit may be damaged. Contact the service center.
0002	Axis name	Sfty obsrvation: Cmd spd err The speed exceeding the speed set with the parameter was commanded during the speed monitoring mode. The name of the axis with an error is displayed.	Check the speed monitoring parameter and the user PLC. Restart the NC.
0003	Axis name	Sfty obsrvation: FB pos err The commanded position transmitted to the servo drive unit from NC and the feedback position to be received from the servo drive unit are totally different during the speed monitoring mode. The name of the axis with an error is displayed.	The NC or the servo drive unit may be damaged. Contact the service center.
0004	Axis name	Sfty obsrvation: FB speed err Actual rotation speed of the motor is exceeding the speed set with speed monitoring parameter during the speed monitoring mode. The name of the axis with an error is displayed.	Check the speed observation parameter and the user PLC. Restart the NC.
0005	Door No.	Door signal: Input mismatch Door state signals on the NC side and the drive side do not match. It may be caused by the followings: <ul style="list-style-type: none"> • Cable disconnection • Damaged door switch • Damaged NC or servo drive unit 	Check the cable. Check the door switch. Restart the NC.
0006	Door No.	No spd obsv mode in door open The door open state was detected when the speed monitoring mode was invalid. The causes may be same as the ones for 0005 (Door signal: Input mismatch). Also the user PLC may not be correct.	Check the user PLC. Restart the NC.

Error No.	Alarm No.	Details	Remedy
0007	Axis name	Speed obsv: Para incompatible Two speed monitoring parameters are not matched at the rising edge of the speed monitoring mode signal. The name of the axis with an error is displayed.	Change the relevant parameters so that the two speed monitoring parameters match. Restart the NC.
0008	Contact No.	Contact welding detected Contact welding was detected.	Make sure that contactor's auxiliary b contact signal is output correctly to the device set on "#1380 MC_dp1" and "#1381 MC_dp2". If welding, replace the contactor. Restart the NC.
0009	-	No spec: Safety observation The servo parameter and the spindle parameter of the speed monitor are set for a system with no safety observation option.	Turn OFF the servo parameter SV113/bitF, the spindle parameter SP229/bitF and the spindle type servo parameter SV113/bitF. Then, restart the NC.
0010	-	SDIO connector input volt err 24VDC power is not supplied to SDIO connector correctly. (SDIO 4A pin supply voltage was dropped to 16V or less, or 1ms or more instant power interrupt was detected.) In this case, "Pw sply:Inst pw interpt(DC24V)" alarm occurs because the contactor control output signal cannot be controlled. This state remains until restarting the NC even if the cause of the alarm has been removed.	Check the wiring. Supply 24VDC power to the SDIO connector. Restart the NC.

(12) Class: y21 Safety observation warning

The warning will be cancelled when the cause of the warning is removed.

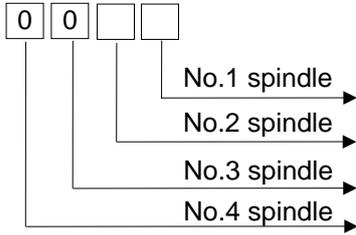
Error No.	Alarm No.	Details	Remedy
0001	Axis name	Speed obsv signal: Speed over The speed exceeds the safety speed limit when the speed monitoring mode signal is ON. The name of the axis with an error is displayed.	When decelerated, the warning will be removed, and the speed monitor will be started.

(13) Class: Y51 Parameter error

An error occurred in a parameter that causes an alarm while the control axis was operating.

Error No.	Details	Remedy
0001	Parameter G0tL illegal The time constant has not been set or the setting exceeded the setting range.	• Check "#2004 G0tL".
0002	Parameter G1tL illegal The time constant has not been set or the setting exceeded the setting range.	• Check "#2007 G1tL".
0003	Parameter G0t1 illegal The time constant has not been set or the setting exceeded the setting range.	• Check "#2005 G0t1".
0004	Parameter G1t1 illegal The time constant has not been set or the setting exceeded the setting range.	• Check "#2008 G1t1".
0009	Parameter grid space illegal	• Check "#2029 grspc".
0012	Parameter stapt1-4 illegal The time constant has not been set or the setting exceeded the setting range.	• Check spindle parameters" #3017 stapt1" to "#3020 stapt4".
0015	Parameter skip_tL illegal The time constant has not been set or the setting exceeded the setting range.	• Check "#2102 skip_tL".
0016	Parameter skip_t1 illegal The time constant has not been set or the setting exceeded the setting range.	• Check "#2103 skip_t1".
0017	Parameter G0bdcc illegal "#1205 G0bdcc" for the 2 nd part system is set to acceleration/deceleration before G0 interpolation.	• Check "#1205 G0bdcc".
0018	OMR-II parameter error The OMR-II related parameter settings are incorrect. In this case, the OMR-II is disabled.	• Check the related parameter settings.
0019	PLC indexing stroke length err When the linear axis equal indexing is validated for the PLC indexing axis, "#12804 aux_tleng" has not been set. Otherwise, it is out of the setting range.	• Check "#12804 aux_tleng".
0020	No hi-accu acc/dec t-const ext Option to extend the high-accuracy acceleration/deceleration time constant is unavailable.	• Set "#1207 G1btL" to the value with which high-accuracy time constant extension specification is unavailable.
0101	Values of PC1/PC2 too large The PC1 and PC2 settings used for the rotary axis are too large.	• Check "#2201 PC1" and "#2202 PC2".

(14) Class: Y90 Message: No spindle signal

(Alarm No.) 		<table border="1"> <thead> <tr> <th>Alarm No.</th> <th>Z open phase</th> <th>B open phase</th> <th>A open phase</th> </tr> </thead> <tbody> <tr> <td>1</td> <td></td> <td></td> <td>×</td> </tr> <tr> <td>2</td> <td></td> <td>×</td> <td></td> </tr> <tr> <td>3</td> <td></td> <td>×</td> <td>×</td> </tr> <tr> <td>4</td> <td>×</td> <td></td> <td></td> </tr> <tr> <td>5</td> <td>×</td> <td></td> <td>×</td> </tr> <tr> <td>6</td> <td>×</td> <td>×</td> <td></td> </tr> <tr> <td>7</td> <td>×</td> <td>×</td> <td>×</td> </tr> </tbody> </table>	Alarm No.	Z open phase	B open phase	A open phase	1			×	2		×		3		×	×	4	×			5	×		×	6	×	×		7	×	×	×
Alarm No.	Z open phase	B open phase	A open phase																															
1			×																															
2		×																																
3		×	×																															
4	×																																	
5	×		×																															
6	×	×																																
7	×	×	×																															
Alarm No.	Details	Remedy																																
0001 to 0007	There is an error in the spindle encoder signal. The data transmission to the drive unit is stopped when this error occurs.	<ul style="list-style-type: none"> Check the spindle encoder's feedback cable and the encoder. 																																

6.1.5 System Alarms

(The bold characters are the messages displayed on the screen.)

<p>Z31 <u>DataServer error</u> <u>0000</u></p>	
Warning No.	Explanation
0001	Socket open error(socket)
0002	Socket bind error(bind)
0003	Connection wait queue error(listen)
0004	Connection request error(accept)
0005	Data recv error(socket error)
0006	Data recv error(data error)
0007	Data send error(socket error)
0008	Data send error(data error)
000A	Socket close error(close)

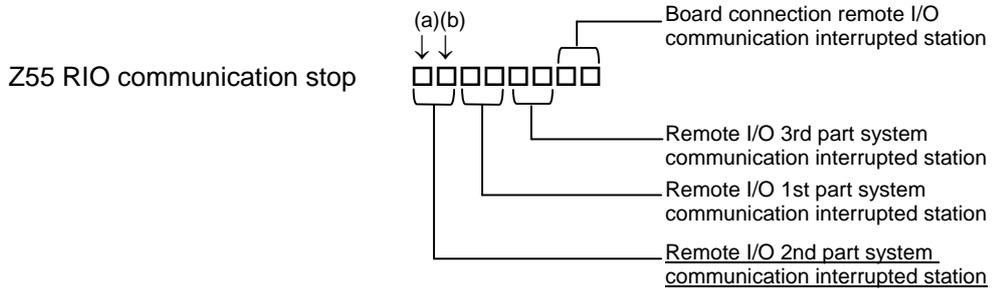
(Note) If warning No. 0001, 0002, 0003, or 000A is displayed, set the parameters, then turn power OFF and turn it ON again.

(The bold characters are the messages displayed on the screen.)

Message	Details	Remedy
Z40 Format mismatch	This appears when the parameter MemVal is formatted at 0, and MemVal is set to 1.	<ul style="list-style-type: none"> • Either return the MemVal setting, or format and restart.
Z51 E2PROM error 00xx	<Type> Z51 E2PROM error 0011: Read error Z51 E2PROM error 0012: Write error	<ul style="list-style-type: none"> • If the same alarm is output by the same operation, the cause is an H/W fault. Contact the Service Center.
Z52 Battery fault 000x	The voltage of the battery inserted in the NC control unit has dropped. (The battery used to save the internal data.) 0001: Battery warning 0002: Battery detecting circuit error 0003: Battery alarm (Note 1)	<ul style="list-style-type: none"> • Replace the battery of the NC control unit. • Check for disconnection of the battery cable. • After treating the battery, check the machining program.
Z53 CNC overheat	The controller or operation board temperature has risen above the designated value. (Note 2)	<ul style="list-style-type: none"> • Cooling measures are required. • Turn OFF the controller power, or lower the temperature with a cooler, etc.
Z55 RIO communication stop	This occurs when an error occurs in the communication between the controller and remote I/O unit. <ul style="list-style-type: none"> • Cable breakage • Remote I/O unit fault • Power supply to remote I/O unit fault (Note 3)	<ul style="list-style-type: none"> • Check and replace the cables. • Replace the remote I/O unit. • Check the power supply. (existence of supply, voltage)
Z57 System warning	The program memory capacity setting value cannot be formatted. The expansion cassette (HR437) is not mounted after formatting. An expansion cassette different from the expansion cassette (HR437) mounted during formatting is mounted.	Check the state of the following items. <ul style="list-style-type: none"> • Program memory capacity • Status of expansion cassette (HR437) mounting • APLC open option
Z58 ROM write not completed	The machine tool builder macro program was not written to the FROM after being registered, edited, copied, condensed, merged, the number changed, or deleted.	<ul style="list-style-type: none"> • Write the machine tool builder macro program to the FROM. • If the operations, such as editing, done while the NC power was OFF can be invalidated, the program does not need to be written to the FROM.
Z59 Acc/dec time cnst too large	Acceleration and deceleration time constants are too large. (This alarm is output at the same time as "T02 Acc/dec time cnst too large 0206.")	<ul style="list-style-type: none"> • Increase the value specified as the "#1206 G1bF" parameter. • Decrease the value specified as the "#1207 G1btL" parameter. • Lower the feedrate.
Z60 Fieldbus communication error	Communication error has occurred on the Fieldbus communication using HN571/HN573/HN575.	<ul style="list-style-type: none"> • Refer to (Note 4) for details.
Z64 Valid term soon to be expired xx	The valid term will be expired in less than a week. Remaining valid term is xx more days.	<ul style="list-style-type: none"> • Obtain a decryption code by making a payment. Enter the decryption code.
Z65 Valid term has been expired	No decryption code was input before the valid term was expired.	<ul style="list-style-type: none"> • Obtain a decryption code by making a payment. Enter the decryption code.

(Note 3) RIO communication interrupt

If communication between the control unit and remote I/O unit fails, the alarm and remote I/O unit number are displayed.



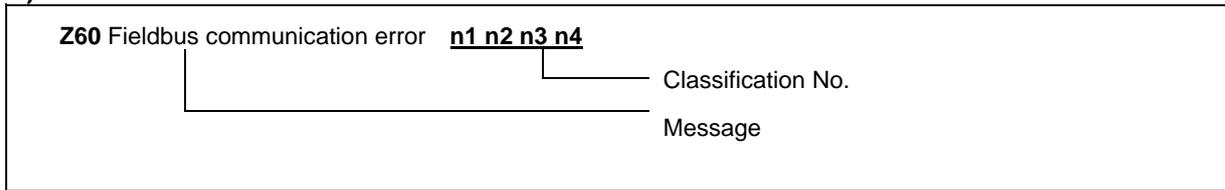
(a) and (b) above indicate the following matters.

Alarm number	RIO (seventh station)	RIO (sixth station)	RIO (fifth station)	RIO (fourth station)
0				
1				X
2			X	
3			X	X
4		X		
5		X		X
6		X	X	
7		X	X	X
8	X			
9	X			X
A	X		X	
B	X		X	X
C	X	X		
D	X	X		X
E	X	X	X	
F	X	X	X	X

Alarm number	RIO (third station)	RIO (second station)	RIO (first station)	RIO (0th station)
0				
1				X
2			X	
3			X	X
4		X		
5		X		X
6		X	X	
7		X	X	X
8	X			
9	X			X
A	X		X	
B	X		X	X
C	X	X		
D	X	X		X
E	X	X	X	
F	X	X	X	X

This also applies for the remote I/O 1st part system communication interrupted station, remote I/O 3rd part system communication interrupted station and board connection remote I/O communication interrupted station.

(Note 4)



Class. No.	Details																		
n1	Shows state of the master channel (shown in hexadecimal number)																		
	00: Offline In initializing																		
	40: Stop Cutting I/O communication																		
	80: Clear Resetting output data of each slave by sending 0 data.																		
	C0: In operation In I/O communication																		
n2	Shows error state (shown in hexadecimal number)																		
	Bit 7 6 5 4 3 2 1 0 <input type="checkbox"/> <input type="checkbox"/>																		
	<table border="1" style="width: 100%; border-collapse: collapse;"> <thead> <tr> <th style="text-align: center;">BIT</th> <th style="text-align: center;">Details</th> </tr> </thead> <tbody> <tr> <td style="text-align: center;">0</td> <td>Control error: Parameter error</td> </tr> <tr> <td style="text-align: center;">1</td> <td>Auto clear error: Communication with all the slave channels was cut because a communication with one slave channel had an error.</td> </tr> <tr> <td style="text-align: center;">2</td> <td>Non exchange error: Slave channel with communication error is found</td> </tr> <tr> <td style="text-align: center;">3</td> <td>Fatal error: The communication cannot be continued because sever network failure exists.</td> </tr> <tr> <td style="text-align: center;">4</td> <td>Event error: Short-circuit was found on the network.</td> </tr> <tr> <td style="text-align: center;">5</td> <td>Not ready: CNC communication is not ready.</td> </tr> <tr> <td style="text-align: center;">6</td> <td>Time out error: Time out was detected in communication with each channel.</td> </tr> <tr> <td style="text-align: center;">7</td> <td>Not used</td> </tr> </tbody> </table>	BIT	Details	0	Control error: Parameter error	1	Auto clear error: Communication with all the slave channels was cut because a communication with one slave channel had an error.	2	Non exchange error: Slave channel with communication error is found	3	Fatal error: The communication cannot be continued because sever network failure exists.	4	Event error: Short-circuit was found on the network.	5	Not ready: CNC communication is not ready.	6	Time out error: Time out was detected in communication with each channel.	7	Not used
	BIT	Details																	
	0	Control error: Parameter error																	
	1	Auto clear error: Communication with all the slave channels was cut because a communication with one slave channel had an error.																	
	2	Non exchange error: Slave channel with communication error is found																	
	3	Fatal error: The communication cannot be continued because sever network failure exists.																	
	4	Event error: Short-circuit was found on the network.																	
	5	Not ready: CNC communication is not ready.																	
6	Time out error: Time out was detected in communication with each channel.																		
7	Not used																		
n3	Shows error No. (shown in hexadecimal number) Refer to "(a) Error in master channel" and "(b) Error in slave channel" for details.																		
n4	Shows slave No. where communication error has occurred. (shown in hexadecimal number) "FF" means an error in master channel.																		

(a) Error in master channel (when remote address with an error is FF (hexadecimal number))

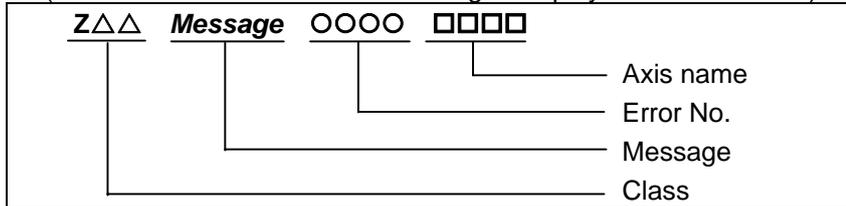
Value in n3	Details	Remedy
0	No error	Operating normally
32	No USR_INTF-task	Damage in HN571 . Replace HN571 .
33	No global data field	
34	No FDL-task	
35	No PLC-task	
37	Master parameter incorrect	
39	Slave parameter incorrect	
3C	Data offset exceeding allowable set value received	Check the configuration setting.
3D	Slave data send range overlap	
3E	Slave data receive range overlap	
3F	Not set data hand shake	Damage in HN571 . Replace HN571 .
40	RAM range exceeded	
41	Slave parameter data set illegal	
CA	No segment	
D4	Data base read illegal	Download the configuration data again.
D5	Operating system illegal	Damage in HN571 . Replace HN571 .
DC	Watch dog error	
DD	Hand shake mode No data communication by 0	
DE	Master auto clear mode	When setting auto clear mode, the auto clear mode was performed because one slave was not able to connect in run time.

(b) Error in slave channel (when remote address with an error is other than FF (hexadecimal number))

Value in n4	Details	Remedy
2	Station overflow reported	Check the configuration of slave channel in which error has occurred. Check if there is any short-circuit in wire to bus.
3	Station stopped responding to master command	
9	No slave required responding data	
11	No station respond	
12	No master to logical token ring	
15	Illegal parameter requested	

6.1.6 Absolute Position Detection System Alarms

(The bold characters are the messages displayed on the screen.)



(1) Class: Z70 Abs data error

This error is displayed if the absolute position data is lost in the absolute position detection system.

Error No.	Details	Remedy	Zero point initialization	Alarm reset when power is turned OFF	Servo alarm No.
0001	Abs posn base set incomplete Zero point initialization is incomplete. Otherwise, the spindle was removed.	Complete zero point initialization.	Required	-	-
0002	Absolute position lost The absolute position reference point data saved in the NC has been destroyed.	Input the parameters. If the reference point data cannot be restored, perform zero point initialization.	(Required)	-	-
0003	Abs posn param changed The parameters used to detect the absolute position have been changed. #1003 iunit #2201 PC1 #1016 iout #2202 PC2 #1017 rot #2218 PIT #1018 ccw #2219 RNG1 #1040 M_inch #2220 RNG2 #2049 type #2225 MTYP	Correctly set the parameters. Turn the power on again, and perform zero point initialization.	Required	-	-
0004	Abs posn initial set illegal The zero point initialization point is not at the grid position.	Reperform zero point initialization.	Required	-	-
0005	Abs posn param restored Restoration was possible with parameter input in the above No.0002 state.	Turn the power on again, and operation will be possible.	Not required	-	-
0080	Abs posn data lost The absolute value data was lost, because the multi-rotation counter data in the detector was incorrect, etc.	Replace the detector and complete zero point initialization.	Required	-	(9E) etc.

Error No.	Details	Remedy	Zero point initialization	Alarm reset when power is turned OFF	Servo alarm No.
0101	Abs posn error(servo alm 25) The power was turned ON again after the servo alarm No. 25 displayed.	Reperform zero point initialization.	Required	-	(25)
0106	Abs posn error(servo alm E3) The power was turned ON again after the servo alarm No. E3 displayed.	Reperform zero point initialization.	Required	-	(E3)

(Note) To release alarm "Z70 Abs data error", enter the parameter data output when establishing the absolute position and turn ON the power again. For the rotary axis, however, the alarm cannot be released by entering the parameter data.

(2) Class: Z71 Abs encoder failure

This alarm is displayed if an error is found in the detector for the absolute position detection system.

Error No.	Details	Remedy	Zero point initialization	Alarm reset when power is turned OFF	Servo alarm No.
0001	AbsEncoder:Backup voltage drop The backup voltage in the absolute position detector dropped.	Replace the battery, check the cable connections, and check the detector. Turn the power ON again, and perform zero point initialization.	Required	- (Z70-0101 displays after power is turned ON again.)	25
0003	AbsEncoder: Commu error Communication with the absolute position detector was not possible.	Check and replace the cables, card or detector. Turn the power ON again, and perform zero point initialization.	(Required) Only when detector is replaced.	Reset	91
0004	AbsEncoder: Abs data changed The absolute position data fluctuated when establishing the absolute position.	Check and replace the cables, card or detector. Turn the power ON again, and perform zero point initialization.	(Required) Only when detector is replaced.	Reset	93
0005	AbsEncoder: Serial data error An error was found in the serial data from the absolute position detector.	Check and replace the cables, card or detector. Turn the power ON again, and perform zero point initialization.	(Required) Only when detector is replaced.	Reset	92
0006	AbsEncoder: Abs/inc posn diffr Servo alarm E3 Absolute position counter warning	Operation is possible until the power is turned off.	(Required) When power is turned ON again.	Reset (Z70-0106 displays after power is turned ON again.)	E3
0007	AbsEncoder: Initial commu err Initial communication with the absolute position detector was not possible.	Check and replace the cables, card or detector. Turn the power ON again, and perform zero point initialization.	(Required) Only when detector is replaced.	Reset	18

(3) Class: Z72 Message: Position check error

This alarm is displayed if an error is detected when comparing the detector's absolute position and controller coordinate values in the absolute position system.

(4) Class: Z73 Message: Absolute position data warning

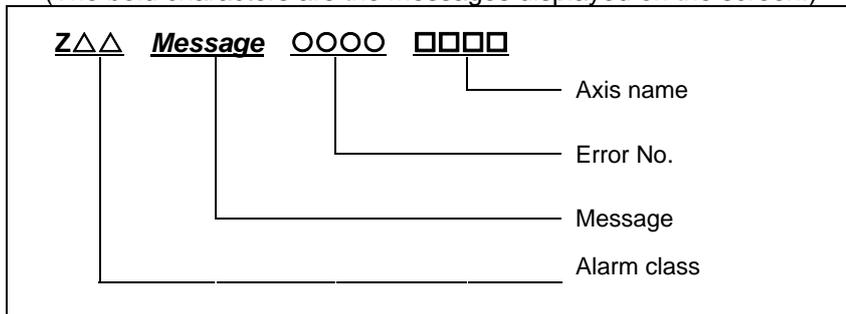
This warning is displayed for the absolute position detection system.

Alarm No.	Details	Remedy
0001	Battery for abs data fault Servo alarm 9F Low battery voltage	If the battery voltage is low or the cable is damaged, there is no need to initialize the absolute position.

(Note) When this alarm occurs, do not turn OFF the drive unit power to protect the absolute position data. Replace the battery while the drive unit power is ON.

6.1.7 Distance-coded Reference Scale Errors

(The bold characters are the messages displayed on the screen.)



(1) Class: Z80 Distance-coded ref scale err

Error No.	Details	Remedy
0001	Basic position lost The basic point data memorized by the NC is broken.	<ul style="list-style-type: none"> Input the parameter. If the basic point data cannot be recovered, perform the initial reference position setting.
0002	Basic position restore The basic point data is recovered by parameter input.	<ul style="list-style-type: none"> Operation can be started after turning the power ON.
0003	No spec: Distance-coded scale Even if the distance-coded reference scale is not included in the specification, it is set to be available.	<ul style="list-style-type: none"> Check the specification. If you do not use this function, set the detector type in servo parameters correctly.

(2) Class: Z81 Synchronous control

Error No.	Details	Remedy
0001	R-pos adjustment data lost Reference position adjustment value data in the NC is damaged.	<ul style="list-style-type: none"> Input the parameter. If the data cannot be recovered by the parameter, establish the reference position again.
0002	R-pos adjustment data restored After the error 0001, by inputting the parameter, the data was recovered.	<ul style="list-style-type: none"> After the reference position establishment, you can continue the operation.

6.1.8 Messages during Emergency Stop

(The bold characters are the messages displayed on the screen.)

<p>EMG Emergency stop □□□□</p> <p style="margin-left: 100px;">└─Error items</p>

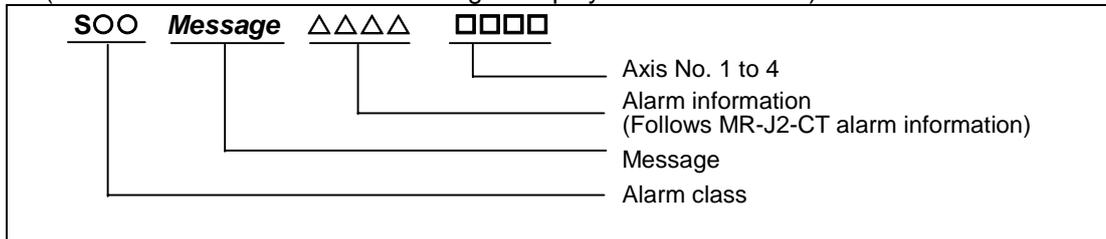
Error Item	Details	Remedy
PLC	The user PLC has entered the emergency stop state during the sequence process.	<ul style="list-style-type: none"> Investigate and remove the cause of the user PLC emergency stop.
EXIN	The emergency stop input signal for machine operation board or handy terminal is significant (open).	<ul style="list-style-type: none"> Cancel the emergency stop input signal. Check the wiring to see if any wiring is broken.
SRV	An alarm occurred in the servo system causing an emergency stop.	<ul style="list-style-type: none"> Investigate and remove the cause of the servo alarm.
STOP	The user PLC (ladder sequence) is not running.	<ul style="list-style-type: none"> Check if the rotary switch CS2 on the top of the controller front panel is set to 1. Check if the PLC edit file save screen (onboard function) [4RUN/SP] (run/stop) switch is turned ON.
SPIN	Spindle drive unit not mounted The spindle drive unit is not mounted.	<ul style="list-style-type: none"> Cancel the causes of the other emergency stop. Check emergency stop signal input in the spindle drive unit.
PC_H	High-speed PC processing abnormal	<ul style="list-style-type: none"> Check the sequence program. (To stop monitoring the high-speed PC processing temporarily, set 1 in "#1219 aux03/bit1". Disable the monitoring function only as a temporary measure.)
PARA	Setting of the door open II fixed device is illegal. The dog signal random assignment parameter setting is illegal.	<ul style="list-style-type: none"> Specify the "#1155 DOOR_m" and "#1156 DOOR_s" parameters correctly. (When the door open II fixed device is not used, set "#1155 DOOR_m" and "#1156 DOOR_s" to "100".) Correctly set the "#2073 zrn_dog", "#2074 H/W_OT+", "#2075 H/W_OT-" and "#1226 aux10/bit5" parameters.

Error No.	Details	Remedy
LINK	If the FROM/TO instruction is not executed within 500 ms, an emergency stop occurs.	<ul style="list-style-type: none"> Try to execute the FROM or TO instruction one or more times every 500 ms. * Measure the time in which no interrupt request is issued from MELSEC and store the result in the R register. R10190: Current time-out counter R10191: Counter for maximum time-out after power-on R10192: Counter for maximum time-out after system start-up (backed up)
	MELSEC is held in error and reset states.	<ul style="list-style-type: none"> Check the MELSEC states.
	The contents of MELSEC-specific code area in buffer memory have been destroyed.	<ul style="list-style-type: none"> Check the MELSEC states.
	PLC serial link communication has stopped. (Note) When "WAIT" is entered for the PLC serial link, only the preparation sequence has been established before the communication stops. Therefore, it is supposed that the basic specification parameters related to serial link parameters #1902 and #1903 are incorrect or the #1909 set-time is too short.	<ul style="list-style-type: none"> Check that HR571 card wiring and external sequencer transmission are normal. Check the diagnostic screen for link communication errors. Check whether the basic specification parameters related to serial link parameters are specified correctly.
WAIT	The preparation sequence is not sent from the master station. Otherwise, the contents of the received preparation sequence are inconsistent with those of the parameters, so that the usual sequence cannot be started. (Note) When "LINK" is also entered for the PLC serial link, refer to "Note" in the section, "LINK".	<ul style="list-style-type: none"> Check that the HR571 card rotary switch and wiring and the external sequencer transmission are normal. Check the diagnostic screen for link communication errors.
XTEN	The HR571 card operates abnormally or the rotary switch is set incorrectly.	<ul style="list-style-type: none"> Check the HR571 card rotary switch and replace the HR571 card if required.
LAD	The user PLC (ladder sequence) has an illegal code.	<ul style="list-style-type: none"> Check the user PLC (ladder sequence) to see if it uses illegal device numbers or constants.
CVIN	The external emergency stop function for power supply is valid. So, the emergency stop input signal for power supply is significant (open).	<ul style="list-style-type: none"> Cancel the emergency stop input signal. Check the wiring to see if any wiring is broken.
MCT	An emergency stop occurs because the contactor shutoff test is executing.	<ul style="list-style-type: none"> Automatically cancel the emergency stop after the contactor shutoff is confirmed. When the contactor shutoff cannot be confirmed within 5 seconds after contactor shutoff test signal (Y742) is input, "Y20 Contactor welding detected" alarm will occur, and the emergency stop status remains. Turn the power ON again after confirming "contactor's auxiliary b contact" signal is correctly output to the device that is set with "#1330 MC_dp1" and "#1331 MC_dp2".

6.1.9 Auxiliary Axis Alarms

Refer to "1.3 Servo/spindle alarms" for details of the alarm class and alarm clear class combination.

(The bold characters are the messages displayed on the screen.)



(1) Class: S01 Aux ax servo alarm

Alarm information	Details	Remedy
0011	Aux ax PCB err (Drive circuit) An error occurred in the drive unit's internal PCB.	<ul style="list-style-type: none"> Replace servo drive unit.
0013	Aux ax S/W processing error An error occurred in the drive unit's internal reference clock.	<ul style="list-style-type: none"> Replace servo drive unit.
0016	Aux ax motor/detector type err	
	Motor type error.	<ul style="list-style-type: none"> Use a correct drive unit and motor combination.
	Detector initial communication error.	<ul style="list-style-type: none"> Connect correctly. Replace the motor. Replace or repair cable.
	Detector CPU error.	<ul style="list-style-type: none"> Replace the motor (detector).
0017	Aux ax PCB error(A/D err) An error occurred in the drive unit's internal A/D converter.	<ul style="list-style-type: none"> Replace servo drive unit.
0025	Aux ax absolute posn lost An error occurred in the detector's internal absolute position data.	<ul style="list-style-type: none"> Turn the power ON for 2 to 3 minutes while the alarm is occurring, and then turn the power ON again. Replace the battery, and initialize the absolute position again.
0034	Aux ax CRC error An error occurred in the communication with the NC.	<ul style="list-style-type: none"> Take countermeasures against noise.
0036	Aux ax communication timeout Communication with the NC was cut off.	<ul style="list-style-type: none"> Connect correctly. Turn the NC power ON. Replace the drive unit or NC.
0037	Aux ax parameter error The parameter setting value is incorrect.	<ul style="list-style-type: none"> Set the parameter correctly.
0038	Aux ax frame error An error occurred in the communication with the NC.	<ul style="list-style-type: none"> Take countermeasures against noise.
0039	Aux ax commu INFO error Undefined data was transferred from the NC.	<ul style="list-style-type: none"> Change the NC software version to a compatible version.

(2) Class: S02 Aux ax servo alarm

Alarm information	Details	Remedy
0011	Aux ax PCB err (Drive circuit) An error occurred in the drive unit's internal PCB.	<ul style="list-style-type: none"> Replace servo drive unit.
0013	Aux ax S/W processing error An error occurred in the drive unit's internal reference clock.	<ul style="list-style-type: none"> Replace servo drive unit.
0015	Aux ax EEROM error A write error occurred to the EEROM in the drive unit.	<ul style="list-style-type: none"> Replace servo drive unit.
0017	Aux ax PCB error(A/D err) An error occurred in the drive unit's internal A/D converter.	<ul style="list-style-type: none"> Replace servo drive unit.
0018	Aux ax PCB error(LSI err) An error occurred in the drive unit's internal LSI.	<ul style="list-style-type: none"> Replace servo drive unit.
0020	Aux ax detector error An error occurred in the communication between the servo drive unit and detector.	<ul style="list-style-type: none"> Connect correctly. Replace or repair cable.
0024	Aux ax ground fault detection A ground fault of the output was detected when the power was turned ON.	<ul style="list-style-type: none"> Repair the ground fault section. Replace the cable or motor.

(3) Class: S03 Aux ax servo alarm

Alarm information	Details	Remedy
0010	Aux ax under voltage The power voltage is 160V or less.	<ul style="list-style-type: none"> Review the power supply. Replace the servo drive unit.
0030	Aux ax regeneration error The tolerable regeneration power of the internal regenerative resistor or external regenerative option was exceeded.	<ul style="list-style-type: none"> Set the parameter #50002 correctly. Connect correctly. Lower the positioning frequency. Change the regenerative option to a larger capacity. Lower the load. Review the power supply.
	Regenerative transistor error	<ul style="list-style-type: none"> Replace the servo drive unit.
0031	Aux ax overspeed The motor's rotation speed exceeded the tolerable momentary speed.	<ul style="list-style-type: none"> Increase the acceleration/ deceleration time constant. Review the gear ratio. Replace the detector.
0032	Aux ax overcurrent A current exceeding the servo drive unit's tolerable current flowed.	<ul style="list-style-type: none"> Repair the wiring. Replace the servo drive unit. Take countermeasures against noise.

Alarm information	Details	Remedy
0033	Aux ax overvoltage The voltage of the converter in the servo drive unit was 400V or more.	<ul style="list-style-type: none"> • Wire correctly. • Replace the servo drive unit. • For the internal regenerative resistor, replace the drive unit. • For the external regenerative option, replace the regenerative option.
0046	Aux ax motor overheating An operation state causing the motor to overheat continued.	<ul style="list-style-type: none"> • Reduce the motor load. • Review the operation pattern.
0050	Aux ax overload 1 The servo drive unit or servomotor overload protection function activated.	<ul style="list-style-type: none"> • Reduce the motor load. • Review the operation pattern. • Change to a motor or drive unit with large output. • Change the setting of the automatic tuning response characteristics. • Correct the connection. • Replace the servomotor.
0051	Aux ax overload 2 The max. output current flowed for several seconds due to a machine collision or overload.	<ul style="list-style-type: none"> • Review the operation pattern. • Change the setting of the automatic tuning response characteristics. • Correct the connection. • Replace the servomotor.
0052	Aux ax excessive error A position deflection exceeding the excessive error detection setting value occurred.	<ul style="list-style-type: none"> • Increase the acceleration/ deceleration time constant. • Increase the torque limit value. • Review the power facility capacity. • Review the operation pattern. • Replace the servomotor. • Connect correctly. • Repair or replace the cable.

(4) Class: S52 Message: Aux ax servo warning

Alarm information	Details	Remedy
0092	Aux ax battery voltage drop The absolute position detection battery voltage dropped.	<ul style="list-style-type: none"> • Mount a battery. • Replace the battery and initialize the absolute position.
00E0	Aux ax overregeneration warning The regeneration power may have exceeded the tolerable range of the built-in regenerative resistor or external regenerative option.	<ul style="list-style-type: none"> • Lower the positioning frequency. • Change the regenerative option to a larger one. • Lower the load.
00E1	Aux ax overload warning The overload alarm 1 could occur.	<ul style="list-style-type: none"> • Refer to the items for S03 0050.
00E3	Aux ax abs position counter warning There is an error in the absolute position detector internal data.	<ul style="list-style-type: none"> • Take countermeasures against noise. • Replace the servomotor.
00E9	Aux ax main circuit OFF warning The servo ON signal was input while the main circuit power was OFF. The contactor operation is faulty.	<ul style="list-style-type: none"> • Turn ON the main circuit power.

(5) Class: Z70 Message: Aux ax abs position error

Alarm information	Details	Remedy
0001	Aux ax abs posn base set incomplete The zero point (reference point) has not been initialized in the absolute position system.	<ul style="list-style-type: none"> • Initialize the zero point (reference point).
0002	Aux ax absolute position lost The absolute position coordinate data in the drive unit has been lost.	<ul style="list-style-type: none"> • Initialize the zero point (reference point).
0003	Aux ax abs posn param changed The absolute position system related parameters have been changed or lost.	<ul style="list-style-type: none"> • Correctly set the parameters and then initialize the zero point (reference point).

(6) Class: Z71 Message: Aux ax drop voltage

Alarm information	Details	Remedy
0001	Aux ax abs encoder: back up voltage drop The data in the detector has been lost. Battery voltage drop. Detector cable wire breakage or looseness.	<ul style="list-style-type: none"> • Check the battery and detector cable and then initialize the zero point (reference point).

(7) Class: Z73 Message: Aux ax system warning

Alarm information	Details	Remedy
0001	Aux ax battery for abs data fault Battery voltage drop. Detector cable wire breakage or looseness.	<ul style="list-style-type: none"> Check the battery and detector cable. The zero point does not need to be initialized.
0003	Aux ax absolute position counter warning An error occurred in the detector's absolute position counter.	<ul style="list-style-type: none"> Replace the detector.

(8) Class: M00 Aux ax operation error

Alarm information	Details	Remedy
0001	Aux ax dog overrun When executing dog-type reference position, the zero point return speed is too fast or the dog length is too short.	<ul style="list-style-type: none"> Lower the zero point return speed or increase the dog length.
0003	Aux ax R-pnt direction illegal When executing reference position return, the axis was moved in the opposite of the designated direction.	<ul style="list-style-type: none"> Move the axis in the correct direction.
0004	Aux ax external interlock The axis interlock function is valid.	<ul style="list-style-type: none"> Cancel the interlock signal
0005	Aux ax internal interlock An interlock was established by the servo OFF function.	<ul style="list-style-type: none"> Cancel the servo OFF.
0007	Aux ax soft limit The soft limit was reached.	<ul style="list-style-type: none"> Check the soft limit setting and machine position
0024	Aux ax R ret invld at abs alm Reference position return was executed during an absolute position alarm.	<ul style="list-style-type: none"> Initialize the absolute position reference point and then fix the absolute position coordinates.
0025	Aux ax R ret invld at ini Reference position return was executed while initializing the absolute position.	<ul style="list-style-type: none"> Initialize the absolute position reference point and then fix the absolute position coordinates.

(9) Class: M01 Aux ax operation error

Alarm information	Details	Remedy
0101	Aux ax no operation mode The operation mode is not designated, or the operation mode was changed during axis movement.	<ul style="list-style-type: none"> • Correctly designate the operation mode.
0103	Aux ax feedrate 0 The operation parameter's feedrate setting is zero. The operation parameter feedrate setting is zero. Or, the override is valid, and the override value is zero.	<ul style="list-style-type: none"> • Set a value other than zero in the feedrate setting or override value.
0160	Aux ax sta No. illegal A station No. exceeding the No. of indexed divisions was designated.	<ul style="list-style-type: none"> • Correctly designate the station No.
0161	Aux ax R-pnt ret incomplete Automatic/manual operation was started before reference position return was executed with the incremental system.	<ul style="list-style-type: none"> • Execute the reference position return.
0162	Aux abs position initializing The start signal was input while initializing the absolute position reference point.	<ul style="list-style-type: none"> • Complete the absolute position reference point initialization.
0163	Aux ax abs position error The start signal was input during an absolute position alarm.	<ul style="list-style-type: none"> • Initialize the absolute position reference point and then fix the absolute position coordinates.
0164	Aux ax arbitrary positioning The manual operation mode was started during the random positioning mode.	<ul style="list-style-type: none"> • Turn the random positioning mode OFF before switching to the manual operation mode.
0165	Aux uneven index sta No. ilg The commanded station No. was higher than 9 or the number of indexing stations during uneven indexing.	<ul style="list-style-type: none"> • Check the commanded station No. and the parameter "#50100 station" setting.

(10) Class: Y02 Auxiliary axis MCP alarms

An error occurred during data transfer between the MCP and auxiliary axis drive unit after turning on the power.

Error No.	Details		Remedy
0050	Aux ax sys alm: Proc time over		The software or hardware may be damaged. Contact the service center.
0051	0000	Aux ax commu er: CRC error 1 (10 times/910.2ms)	A communication error has occurred between the controller and drive unit. <ul style="list-style-type: none"> • Take measures against noise. • Check that the communication cable connector between the controller and drive unit and one between the drive unit are tight. • Check whether the communication cable between the controller and drive unit and one between the drive units are disconnected. • A driving drive unit may be faulty. Take a note of the 7-segment LED contents of each driving drive unit and report to the Service Center.
	0001	Aux ax commu er: CRC error 2 (2 continuous times)	
	0002	Aux ax commu er: Recv timing (2 continuous times)	
	xx03	Aux ax commu er: Data ID (2 continuous times) xx: Axis No.	
	xx04	Aux ax commu er: Recv frame no. (2 continuous times) xx: Axis No.	

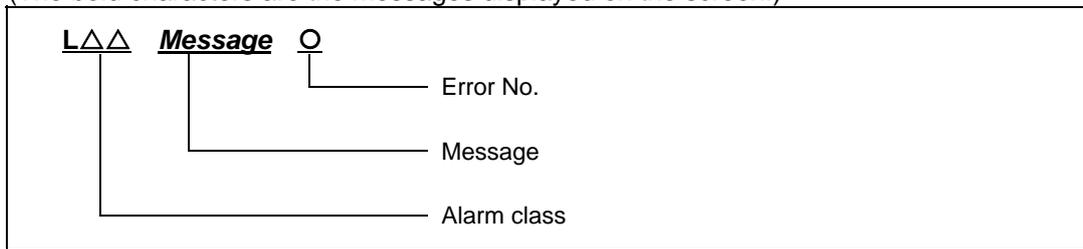
(11) Class: Y03 Message: Aux ax drive unit unequipped

The drive unit is not properly connected.

Error No.	Details	Remedy
Axis No. 1 to 4	bit correspondence (bit 0: 1st axis, bit 1: 2nd axis, bit 2: 3rd axis, bit 3: 4th axis)	Check the auxiliary axis drive unit mounting state. <ul style="list-style-type: none"> • Check the end of the cable wiring. • Check the cable for broken wires. • Check the connector insertion. The auxiliary axis drive unit input power is not being input. The auxiliary axis drive unit axis No. switch is illegal.

6.1.10 Computer Link Errors

(The bold characters are the messages displayed on the screen.)

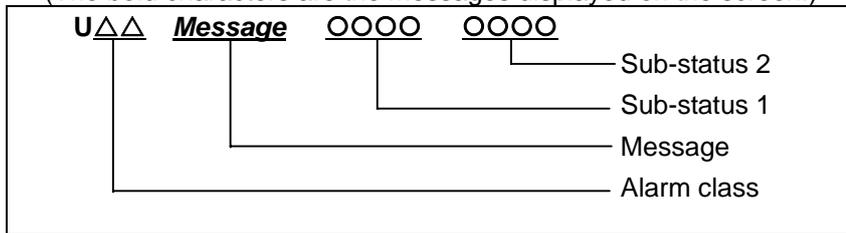


(1) Class: L01 Computer link error

Error No.	Details	Remedy
-2	Serial port being used Serial port has already being opened or cannot be used.	<ul style="list-style-type: none"> • Check whether the same port being used by Anshin-net, etc. • Recheck the parameters for tape operation port.
-4	Timeout error Communication ends with timeout (CNC has a 248-byte receive buffer. The time during which CNC receives 248 bytes exceeds the "TIME-OUT" value set in the I/O device parameter.)	<ul style="list-style-type: none"> • Set a greater timeout value in the input/output device parameter. • Recheck the HOST software as to whether or not the HOST transmits data in response to DC1 from CNC (data request). • Check whether or not start code of computer link parameter is set to 0.
-10	Host ER signal OFF HOST ER (CNC DR) signal is not turned ON.	<ul style="list-style-type: none"> • Check whether or not the cable is disconnected from the connector. • Check whether or not the cable is broken. • Check whether or not the HOST power is turned ON.
-15	Parity H error Communication ends with parity H.	<ul style="list-style-type: none"> • Recheck the HOST software as to whether or not the data to be transmitted to CNC is ISO code.
-16	Parity V error Communication ends with parity V.	<ul style="list-style-type: none"> • Recheck the data to be transmitted to CNC.
-17	Overrun error Although CNC transmits DC3 (request to stop data transfer) to the HOST, it receives data of 10 bytes or more from the HOST, thus terminates communication. When CNC is transmitting data to the HOST, it receives data of 10 bytes or more from the HOST.	<ul style="list-style-type: none"> • Recheck the software as to whether or not the HOST stops transmitting data within 10 bytes after receiving DC3. • Recheck the HOST software as to whether or not the HOST transmits data such as a command or header to CNC during receiving a work program.

6.1.11 User PLC Alarms

(The bold characters are the messages displayed on the screen.)



Message	Sub-status		Details	Remedy
	1	2		
U01 No user PLC	-	-	PLC program is not input. (Note) Emergency stop (EMG) will be applied.	Download the PLC program of the format selected with the PLC environment selection parameters (bit selection #51/bit 4).
U10 Illegal PLC	0x0010	-	PLC scan time error The scan time is 1 second or longer.	Edit the PLC program size to a smaller size.
	0x0040	-	PLC program operation mode illegal PLC program different from the designated mode was downloaded. (Note) Emergency stop (EMG) will be applied.	Download the PLC program having the same format as when the power was reset or turned ON.
	0x0080	-	GPPW ladder code error (Note) Emergency stop (EMG) will be applied.	Download the correct GPPW format PLC program.
	0x008x	-	PLC4B ladder code error An illegal circuit was found in the PLC4B ladder. bit1: PC medium-speed circuit illegal bit2: PC high-speed circuit illegal (Note) Emergency stop (EMG) will be applied.	Download the correct PLC4B format PLC program.
	0x0400	Number of ladder steps	Software illegal interrupt The PLC program process stopped abnormally due to an illegal software command code. (Note) Emergency stop (EMG) will be applied.	Turn the power ON again. If the error is not reset, download the correct PLC program.

Message	Sub-status		Details	Remedy
	1	2		
U10 Illegal PLC	0x800x	Number of PLC program steps	Software exception The PLC program process stopped abnormally due to a bus error, etc.	
			bit 0: BIN command operation error bit 1: BCD command operation error	Refer to the methods for using the BCD and BIN function commands.
			bit6: CALL/CALLS/RET command error bit7: IRET command execution error (Note) Emergency stop (EMG) is applied for bit 6/7.	Turn the power ON again. If the error is not reset, download the correct PLC program.
U50 PLC stopped			The PLC program is stopped.	Start the PLC program.
U55 PLC stopped / is not saved			The PLC program is stopped and not written into ROM.	Write the PLC program into ROM.
U60 Ladder is not saved			The PLC program is not written into ROM.	Write the PLC program into ROM.

(Note) The number of PLC program steps displayed on the screen may not match the actual number of error occurrence steps because of the PLC program timing. Use this as a guideline of the occurrence place.

6.1.12 Network Service Errors

Message	Details	Remedy
N001 Modem initial error	<ul style="list-style-type: none"> There is an error in the modem connection when the power is turned ON. 	<ul style="list-style-type: none"> Check the connection between the NC and modem, connection port and modem power.
N002 Redial over	<ul style="list-style-type: none"> The dial transmission failed more than the designated No. of redial times. 	<ul style="list-style-type: none"> Wait a while, and then transmit again.
N003 TEL unconnect	<ul style="list-style-type: none"> The phone line is not connected. 	<ul style="list-style-type: none"> Check the modem's phone line connection.
N004 Net communication error	<ul style="list-style-type: none"> An error other than the above errors occurred during communication. 	<ul style="list-style-type: none"> Note down the circumstances under which this error occurred, and contact the Service Center.
N005 Invalid net communication	<ul style="list-style-type: none"> The modem connection port is being used for another function such as input/output. The modem connection port settings are incorrect. 	<ul style="list-style-type: none"> Quit using the modem connection port with the other function, and then turn the power ON again. Check the modem connection port settings.
N006 Received result of diagnosis	<ul style="list-style-type: none"> The diagnosis data file has been received 	<ul style="list-style-type: none"> Erase the message.
N007 Send data size over	<ul style="list-style-type: none"> A file larger than Anshin-net server capacity (64Kbyte) has been transmitted during machining data sharing. 	<ul style="list-style-type: none"> Reduce the size of machining program file so that it won't exceed the capacity of Anshin-net server.
N008 No file on server	<ul style="list-style-type: none"> When machining data sharing function is being executed, file reception fails, as the file does not exist on Anshin-net server. 	<ul style="list-style-type: none"> Before receiving file, confirm that a machining program file exists on Anshin-net server.
N009 Password error	<ul style="list-style-type: none"> File reception fails due to wrong password when machining data sharing is being executed. 	<ul style="list-style-type: none"> Input the password again.
N010 Customer number error	<ul style="list-style-type: none"> Data reception fails due to wrong customer number when machining data sharing is being executed. 	<ul style="list-style-type: none"> Input the customer number again.
N011 Storage capacity over	<ul style="list-style-type: none"> As the size of file to be received is bigger than free space on the NC side, file reception fails during machining data sharing. 	<ul style="list-style-type: none"> Ensure sufficient free space on the NC side.
N012 File deletion error	<ul style="list-style-type: none"> A file on Anshin-net server cannot be deleted when machining data sharing is being executed. 	<ul style="list-style-type: none"> Check if the file exists on Anshin-net server. Note down the circumstances under which this error occurred, and contact the Service Center.

6.2 Operation Messages

The following messages display on each screen.

6.2.1 Search-related Operation Messages

Message	Details
Searching	<ul style="list-style-type: none"> The operation search is being executed.
Search completed	<ul style="list-style-type: none"> The operation search was completed correctly.
Verifying	<ul style="list-style-type: none"> The program is being verified.
Verifying stopped	<ul style="list-style-type: none"> The program verification stopped.
Search error	<ul style="list-style-type: none"> Could not find the designated ONB number. Review the ONB number or machining program settings. The operation search could not be completed correctly. When the parameter (#9005) of the tape mode port was set out of the range or when the port was not connected, an operation search of NC serial was performed. The T code list search failed. Review the program name. Could not find the machining program in HD or IC card. Check the parameters for HD operation or IC card operation. Tape search was executed during the HOST LINK communication.
Setting error	<ul style="list-style-type: none"> The directory name exceeded the display range. A directory path for which the entire directory cannot be displayed cannot be designated in the directory name area. The search is not possible because ONB number is not designated.
Restart search is completed	<ul style="list-style-type: none"> The restart search was completed.
Executing restart search	<ul style="list-style-type: none"> The restart search is being executed.
Execute operation search	<ul style="list-style-type: none"> The program is not searched. Execute the operation search.
Can't cancel verify stop	<ul style="list-style-type: none"> The compare stop cannot be canceled because the operation is not in a compare stop.
Executing top search	<ul style="list-style-type: none"> The top search is being executed correctly.
Top search completed	<ul style="list-style-type: none"> The top search was completed correctly.
Top search not completed	<ul style="list-style-type: none"> The restart search (type 2) was executed without the top search.
Can't input data	<ul style="list-style-type: none"> Data input was attempted during M, S, T, B history display.
Input some of ONB	<ul style="list-style-type: none"> Awaiting ONB number input
Verifying stop posn already registered	<ul style="list-style-type: none"> The verify stop position has already been registered.
No searched program	<ul style="list-style-type: none"> The program has not been searched.
Program restarting	<ul style="list-style-type: none"> The program is being restarted.
Program error	<ul style="list-style-type: none"> The restart search was executed in the program with an error.
N/B block not found	<ul style="list-style-type: none"> The restart search was executed, designating N or B No. not exist.
Restart search interrupted by reset	<ul style="list-style-type: none"> Reset was executed during the restart search.
No. of repetitions exceeded	<ul style="list-style-type: none"> The restart search was executed, designating the number of repetitions exceeding the number set in the program.
Program not found	<ul style="list-style-type: none"> The restart search was executed, designating a program No. not exist.

6.2.2 Graphic Display-related Operation Messages

Message	Details
Searching	<ul style="list-style-type: none"> The check search is being executed.
Search error	<ul style="list-style-type: none"> Could not find the designated ONB number. Review the ONB number or machining program settings. The check search could not be completed correctly. When the parameter (#9005) of the tape mode port was set out of the range or when the port was not connected, an operation search of NC serial was performed.
Executing automatic operation	<ul style="list-style-type: none"> An attempt was made to perform operations such as parameter, tool compensation amount data and coordinate system offset settings during auto operation. (Input/output also possible during auto operation) The machining program and MDI data that the operator is attempting to edit cannot be edited during auto operation. It is not possible to be executed a check search or check start-up during auto operation. An attempt was made to set the verify stop position during auto operation.
Setting error	<ul style="list-style-type: none"> The directory name exceeded the display range. A directory path for which the entire directory cannot be displayed cannot be designated in the directory name area. The check search is not possible because ONB number is not designated. A non-existent axis name or setting that does not exist in the format was made when making the display mode settings. Set using an existing axis name. Incorrect tool No. was specified at the execution of tool clear.
Execute operation search	<ul style="list-style-type: none"> The program is not searched. Execute the operation search.
Checking	<ul style="list-style-type: none"> The program check is being executed.
Check stopped	<ul style="list-style-type: none"> The program check is being stopped.
Work form is illegal	<ul style="list-style-type: none"> A workpiece is not drawn correctly because the two or more workpiece widths are set to "0".
Push [Check reset] menu	<ul style="list-style-type: none"> An error occurred in the graphic check. Press the [Check reset] menu key to reset the error.
Executing trace	<ul style="list-style-type: none"> The trace mode is valid.
Program check completed	<ul style="list-style-type: none"> The trace in the program check completed with an M02/M30 code.
Reset complete	<ul style="list-style-type: none"> The program check was reset.
Tool interfere with work	<ul style="list-style-type: none"> The tool contacted the workpiece when performing rapid traverse (G0) movement with the interference check enabled.
Program checking	<ul style="list-style-type: none"> The program check is being executed.
Executing reset operation	<ul style="list-style-type: none"> The program check is being reset.
Executing trace(tip posn)	<ul style="list-style-type: none"> The tip position trace mode is valid.
Executing BG simulation	<ul style="list-style-type: none"> The back ground simulation is being executed.
Draw library inside error (n)	<ul style="list-style-type: none"> Contact the service center.

Message	Details
Program check not possible	• Graphic check was attempted during the Computer link B operation.
Execute check search	• The program check is disabled. Execute the check search.
Solid check disabled (memory shortage)	• The 3D program check is disabled due to the memory shortage. Separate the programs and execute the check drawing again. Press the menu [Work init] once before performing the check drawing again.

6.2.3 Variable (Common variables, local variables) - related Operation Messages

Message	Details
Erase? (Y/N)	• Message to confirm the line erase. [Y] or [INPUT] : Variables are deleted. [N] : Variables are not deleted.

6.2.4 PLC Switch-related Operation Messages

Message	Details
Set up parameter?(Y/N)	• Message to confirm the parameter setup. [Y] or [INPUT] : It will be possible to make settings. [N] : It will not be possible to make settings.

6.2.5 Compensation-related (Tool compensation, coordinate system offset) Operation Messages

Message	Details
Erase? (Y/N)	• Message to confirm the erasing. [Y] or [INPUT] : Erase the data. [N] : Do not erase the data.
Clear all axes? (Y/N)	• Message to confirm the all axes clear. [Y] or [INPUT] : Clear the data for all axes. [N] : Do not clear the data.
Cannot return to origin	• Operations other than line paste, paste and data input cannot be undone. • It is the initial state. Cannot undo. • The last operation was performed in another part system. Cannot undo the operation.
Clear all? (Y/N)	• Message to confirm the clearing the all data. [Y] or [INPUT] : Clear all data. [N] : Do not clear the data.
Input P number	• The coordinate system [Coord G54.1 P] menu was pressed. • The expansion workpiece coordinate system P No. was input.
Data clear complete	• Clearing of the collection data is complete.
Execute the collection data clear?	• Determines whether to clear the collection data. Press [Y] or [INPUT] to clear the data.

6.2.6 Data Input/Output-related Operation Messages

Message	Details
Overwrite this file?(Y/N)	<ul style="list-style-type: none"> Message to confirm the overwriting. [Y] or [INPUT] : Overwrite the file. [N] : Do not overwrite the file.
Over run error	<ul style="list-style-type: none"> The buffer overran or overflowed.
Memory over	<ul style="list-style-type: none"> The program cannot be written, because the memory capacity will be exceeded.
Edit lock B	<ul style="list-style-type: none"> It is not possible to change machining program B (8000 to 8999: user standard subprogram) or machining program C (9000 to 9999: machine tool builder custom program) as edit lock B is enabled.
Edit lock C	<ul style="list-style-type: none"> It is not possible to change machining program C (9000 to 9999: machine tool builder custom program) as edit lock C is enabled.
Can't make directory on this device	<ul style="list-style-type: none"> Creation of a directory was attempted for a device that cannot have a directory.
Designated file does not exist	<ul style="list-style-type: none"> The file specified in device A, and B does not exist. The applicable file does not exist in the specified directory.
The file name is a directory	<ul style="list-style-type: none"> A directory was designated for the file transfer. A directory cannot be transferred.
Change complete	<ul style="list-style-type: none"> The data conversion completed correctly.
Changing	<ul style="list-style-type: none"> The data is being converted.
Erase complete	<ul style="list-style-type: none"> The file has been erased.
Erase ended. Some file not erased	<ul style="list-style-type: none"> The file erasing completed, but there are some files that could not be erased.
Verify error	<ul style="list-style-type: none"> An error occurred when performing a file verification.
Compare error. Compare next file?(Y/N)	<ul style="list-style-type: none"> Message to confirm the comparison [Y] or [INPUT] : Compare the next file. [N] : Do not compare the next file.
Compare complete	<ul style="list-style-type: none"> The data comparison completed.
Verifying	<ul style="list-style-type: none"> The data is being compared.
The file already exists	<ul style="list-style-type: none"> The input file name already exists. The file name after renaming already exists.
Can't erase designated file	<ul style="list-style-type: none"> Erasing was attempted of a file that cannot be erased.
Can't rename designated file	<ul style="list-style-type: none"> An attempt was made to change the name of a file that cannot be renamed.
Can't condense designated file	<ul style="list-style-type: none"> Condensing of a file that cannot be condensed was attempted.
Designated file is locked	<ul style="list-style-type: none"> Changing was attempted of a locked file.
Can't open file for dev A	<ul style="list-style-type: none"> Could not find the file for device A. Or, the file is in a state in which it cannot be accessed.
Can't open file for dev B	<ul style="list-style-type: none"> Could not find the file for device B. Or, the file is in a state in which it cannot be accessed.
Can't read file for dev A	<ul style="list-style-type: none"> Could not read in the file for device A. Recheck the connection status for device A or the input/output parameter setting.
Can't read file for dev B	<ul style="list-style-type: none"> Could not read in the file for device B. Recheck the connection status for device B or the input/output parameter setting.

Message	Details
Can't close file for dev A	• Contact the service center
Can't close file for dev B	• Contact the service center.
Can't write file for dev A	• Could not write in the file for device B. Recheck the connection status for device A or the input/output parameter setting.
Can't write file for dev B	• Could not write in the file for device B. Recheck the connection status for device B or the input/output parameter setting.
Can't seek file for dev A	• Contact the service center.
Can't seek file for dev B	• Contact the service center.
File name not designated for dev A	• A file name was not designated for device A.
File name not designated for dev B	• A file name was not designated for device B.
Can't open directory for dev A	• Could not find a directory corresponding to device A.
Can't open directory for dev B	• Could not find a directory corresponding to device B.
Different devices designated in A and B	• The same device must be designated for devices A and B, but differing devices were designated.
Timeout error	• A timeout error occurred when communicating with the external device.
Checking	• Cannot be executed during a check
Make directory complete	• Creation of the directory has been completed.
Dir create Complete	• Creation of the directory has been completed.
Can't make directory	• An error occurred while creating the directory.
The directory is not empty	• A file was found in the directory. Erase the file in the directory.
Directory pass is illegal	• The designated directory path name is illegal. Input a correct directory path name.
Data protect	• Setting, erasing, parameter setting, etc., of the various data is prohibited, because the data protect key is validated.
Transfer complete	• The data transfer completed correctly.
Transferring	• The data is being transferred.
Parity H error	• A parity H error was detected.
Parity V error	• A parity V error was detected.
File entry over	• The No. of registration files designated in the specifications was exceeded, so the file could not be registered.
Program No. not found in the file	• There is no program number in the selected file.
Executing format	• The formatting is being executed.
Format complete	• The formatting completed.
Format error	• The formatting failed.
Framing error	• An error occurred between the NC and the external device.
Variable conversion error	• An error occurred during the M2 macro conversion, and the conversion failed.

Message	Details
Merge complete	• The data merge completed.
Merge execution	• The data merge is being executed.
Memory alloc error	• Securing of the communication data range failed.
Rename complete	• The rename has been completed.
OK? (Y/N)	• Message to confirm the operation. [Y] or [INPUT] : Execute the operation [N] : Cancel the operation.
I/O not ready	• An error occurred between the NC and the external device.
I/O parameter error	• The external device settings and input/output parameters do not match.
I/O port busy	• Input/output was not possible as the I/O port is busy.
FD write protect	• The FD is write-protected. Release the write protection.
PLC running	• An attempt was made to input a user PLC during PLC RUN. • Stop the PLC on the maintenance screen.
FD not ready	• An attempt was made to perform an FD operation search with no FD. • An attempt was made to display the FD list with no FD.
MemoryCard not ready	• An attempt was made to perform operations with no memory card.
DS not ready	• An attempt was made to perform operations with no data server.
Can't write in READ-ONLY file	• Contact the service center.
Condense complete	• Condensing has been completed.
A directory does not exist	• The specified directory does not exist.
Setting complete normally	• Decryption code setting file of the credit system was set normally.

6.2.7 Parameter-related Operation Messages

Message	Details
Designate copy end posn	<ul style="list-style-type: none"> The copy start position was specified using the cursor. Continue and specify the copy end position using the cursor.
Copy start posn and end posn reversed	<ul style="list-style-type: none"> When the copy range was designated, a position before the start position was designated as the end position.
Columns of copy start and end different	<ul style="list-style-type: none"> A different column (axis or part system) was specified for the copy start/end position at the screen with the arrangement configuration for each axis and part system.
Copy range is inadequate	<ul style="list-style-type: none"> Could not find the parameter No. for the copy start position. A value larger than the last parameter No. was designated as the copy end position parameter No. Check the designated copy range.
Setting error	<ul style="list-style-type: none"> The port is already being used. The parameter HOST LINK was turned ON during the Anshin-net communication.
Setting error: column n	<ul style="list-style-type: none"> The nth column setting data is inappropriate when multiple axes were set at the same time (/ division). (Settings have been made up to the (n-1)th column.)
Password is illegal	<ul style="list-style-type: none"> The password designated for displaying the Machine Parameter screen is illegal.
Input the password	<ul style="list-style-type: none"> The menu key for first displaying the Machine Parameter screen was pressed after the power was turned ON.
Paste error	<ul style="list-style-type: none"> An attempt was made to paste in a different parameter from the copy parameter.
Paste? (Y/N)	<ul style="list-style-type: none"> Message to confirm the operation when pasting. [Y] or [INPUT]: Paste the data at the current cursor position. [N] : Do not paste the data.
Data protect	<ul style="list-style-type: none"> Setting, erasing, parameter settings, etc., of the various data is prohibit, because the data protect key is validated.
Executing automatic operation	<ul style="list-style-type: none"> An attempt was made to make parameter settings during auto operation.
Can't select	<ul style="list-style-type: none"> The password designated for displaying the machine parameters has not been input. Display of the machine parameters was attempted on the [Param No.] menu, but the password has not been input. Press the [Machine param] menu, and input the password for displaying the machine parameters.
No odd number for R register start No.	<ul style="list-style-type: none"> An odd number cannot be used for R register start No. Use an even number.

6.2.8 Measurement-related (Workpiece, rotation) Operation Messages

Message	Details
TLM switch OFF	<ul style="list-style-type: none"> Measurement, retrieving of the skip position, writing the coordinates data etc. was attempted when the TLM switch was OFF.
Can't take-in skip posn	<ul style="list-style-type: none"> Could not retrieve the skip position. Check the following: < When measuring the workpiece > <ol style="list-style-type: none"> Is the name of the axis to be measured designated in the basic axis (I, J, K)? Is the axis that executed the axis movement a measurement axis? < When measuring the rotation > <ol style="list-style-type: none"> Is the name of the axis to be measured designated in the coordinate rotation plane (horizontal axis, vertical axis)? Is the axis that executed the axis movement a measurement axis?
Can't write coordinates data	<ul style="list-style-type: none"> Could not obtain the measured axis No. The measured angle was illegal. Could not write into the coordinate system offset. Measure again. The work coordinate offset data was attempted to set when the measurement counter does not have the necessary values. The work coordinate offset data was attempted to set in the slave axis.
Sensor take in not possible	<ul style="list-style-type: none"> Could not retrieve the position measured with the touch sensor. Measure again.
Can't measure	<ul style="list-style-type: none"> Measurement failed.
Sensor signal was illegally turned on	<ul style="list-style-type: none"> The sensor signal was already ON when the tool measurement mode (TLM) signal was validated. The sensor signal turned ON when there was no axis movement after the tool measurement mode (TLM) signal was validated. The sensor signal turned ON at a position within 100μm from the final entry start position. Move the axis in a safe direction after turning the sensor signal OFF or turning the tool measurement mode signal OFF.
Can't write compensation data	<ul style="list-style-type: none"> The cursor position and the cell for writing the compensation amount (length dimension, radius dimension) do not match. Match the cursor position with the cell for writing the compensation amount.
Offset No. not found	<ul style="list-style-type: none"> During manual tool length measurement, the sensor was turned ON designating tool compensation No. not exist. During manual tool length measurement, the measurement method was switched to manual tool length measurement 2 designating tool compensation No. not exist. Correctly set the R register of compensation No.
TLM axis is illegal	<ul style="list-style-type: none"> During 2 or more axes movement, the tool length measurement was executed by turning the sensor ON. Keep the tool away from the sensor and execute the measurement by one axis.
TLM axis not returned to ref. position	<ul style="list-style-type: none"> The tool length measurement was executed by tuning the sensor ON for the axis in which dog-type reference point return has not been executed. Carry out reference point return for the measurement axis.
Can't write rotation parameter	<ul style="list-style-type: none"> The measured result cannot be set in the process parameter.

Message	Details
Can't calculate center & angle for rot	<ul style="list-style-type: none"> • Three points necessary for calculation are not retrieved. • Center shift amount or sensor radius was failed to retrieve. • Calculation of the center and the angle was failed.
Input 0 to coord center & angle for rot	<ul style="list-style-type: none"> • The center and the angle are not "0". Set "0" in the parameter "#8623 Coord rot centr (H)", "#8624 Coord rot centr (V)" and "#8627 Coord rot angle".
Can't calculate	<ul style="list-style-type: none"> • Hole center cannot be determined.
Meas axis not returned to ref. position	<ul style="list-style-type: none"> • The workpiece measurement was executed when dog-type reference point return has not been executed. Carry out reference point return for the measurement axis.

6.2.9 Tool (Tool registration, tool life) -related Operation Messages

Message	Details
Designated group already exists	<ul style="list-style-type: none"> An already existing group No. was designated when changing the group No. (Tool life screen (grp)) An already existing group No. was designated when newly creating the group. (Tool life screen (grp list)) Designate a group No. that does not already exist.
Designated group does not exist	<ul style="list-style-type: none"> Erasing was attempted on the Tool life screen (grp list) of a group that does not exist.
Can't register group	<ul style="list-style-type: none"> The group registration process on the Tool life screen (grp list) failed.
Can't delete group	<ul style="list-style-type: none"> The group deletion process on the Tool life screen (grp list) failed.
Erase? (Y/N)	<ul style="list-style-type: none"> Message to confirm the erasing. [Y] or [INPUT] : Erase the data [N] : Do not erase the data.
Delete all groups? (Y/N)	<ul style="list-style-type: none"> Message to confirm the erasing of all groups. [Y] or [INPUT] : Erase all groups [N] : Do not erase all groups.
Can't delete all groups	<ul style="list-style-type: none"> All groups' deletion cannot be executed because data protection key (KEY 1) is validated or automatic operation mode is validated, etc. during all groups' deletion. All groups' deletion cannot be executed because nothing has been registered to group, also.
Pot number not exist	<ul style="list-style-type: none"> The set pot number does not exist. Check the pot number.
Create new file? (Y/N)	<ul style="list-style-type: none"> Message to confirm the operation when newly creating data, files, etc. [Y] or [INPUT] : Newly create. [N] : Cancel the operation.
Input the tool number	<ul style="list-style-type: none"> Waiting tool number input.
Clear not possible	<ul style="list-style-type: none"> The clear range is incorrect.
Spindle/standby tool display not possible	<ul style="list-style-type: none"> The spindle standby cannot be displayed due to the user PLC setting. Contact the machine tool builder.
Format tool life data? (Y/N)	<ul style="list-style-type: none"> Determines whether to format the tool life management data. When [Y] is input, the formatting is executed.
Tool life format complete	<ul style="list-style-type: none"> The tool life management data formatting completed.
Exists in spindle/standby. Set? (Y/N)	<ul style="list-style-type: none"> An attempt was made to set the same No. as the tool No. for spindle/standby.
Exists in magazine 1. Set? (Y/N)	<ul style="list-style-type: none"> An attempt was made to set the same No. as the tool No. that has been registered to the valid magazine.
Exists in magazine 2. Set? (Y/N)	<ul style="list-style-type: none"> An attempt was made to set the same No. as the tool No. that has been registered to the valid magazine.
Exists in magazine 3. Set? (Y/N)	<ul style="list-style-type: none"> An attempt was made to set the same No. as the tool No. that has been registered to the valid magazine.
Exists in magazine 4. Set? (Y/N)	<ul style="list-style-type: none"> An attempt was made to set the same No. as the tool No. that has been registered to the valid magazine.
Exists in magazine 5. Set? (Y/N)	<ul style="list-style-type: none"> An attempt was made to set the same No. as the tool No. that has been registered to the valid magazine.

6.2.10 Editing-related Operation Messages

Message	Details
Buffer correct not possible	<ul style="list-style-type: none"> • Buffer correction cannot be performed for this program. • Buffer correction was attempted during the BTR operation.
Can't write into file	<ul style="list-style-type: none"> • Could not write the data to the memory with the buffer correction. Contact the service center.
Overwrite this file?(Y/N)	<ul style="list-style-type: none"> • Message to confirm when registering MDI [Y] or [INPUT] : Overwrite the file. [N] : Do not overwrite the file.
Memory over	<ul style="list-style-type: none"> • The program cannot be written, because the memory capacity will be exceeded.
Designated character string not found	<ul style="list-style-type: none"> • Could not find the search results and character string in the program.
Save current file?(Y/N)	<ul style="list-style-type: none"> • Message to confirm the saving. [Y] : Save the changes to the current file. [N] : Do not save the changes to the current file.
A file does not exist	<ul style="list-style-type: none"> • An attempt was made to select and edit a non-existent file.
Executing	<ul style="list-style-type: none"> • Following menu's process is executing now: "Line paste", "Line clear", "Undo", "String search", "String replace", "Miss warning", or "Next miss".
Designated file does not exist	<ul style="list-style-type: none"> • An attempt was made to select and erase a non-existent file.
Erase complete	<ul style="list-style-type: none"> • The data erasing completed.
Can't erase designated file	<ul style="list-style-type: none"> • The selected file cannot currently be erased.
Designated file does not exist	<ul style="list-style-type: none"> • A file that does not exist was designated when file editing.
Designated file already exists	<ul style="list-style-type: none"> • When creating a new file, a file name was designated that already exists.
Replace? (Y/N)	<ul style="list-style-type: none"> • Message to confirm the character string replacement. [Y] or [INPUT] : Replace the character string. [N] : Do not replace the character string.
File access error	<ul style="list-style-type: none"> • Contact the Service Center.
File open error	<ul style="list-style-type: none"> • The designated file is already open.
Editing	<ul style="list-style-type: none"> • A program is being edited on the screen. Save the program to write it into the memory.
Program entry over	<ul style="list-style-type: none"> • The program could not be registered in the memory when attempted, because the No. of registrations designated in the specifications would be exceeded.
Block char nos over	<ul style="list-style-type: none"> • The character number limitation in one block was exceeded.
Paste error	<ul style="list-style-type: none"> • Pasting was attempted within the copy range of the same file.
Copy range is inadequate	<ul style="list-style-type: none"> • The copy range designation is inadequate. Check whether the designated range exists. • The range was designated exceeding 100 lines during mass editing.
Area designation is inadequate	<ul style="list-style-type: none"> • The area designation is inadequate. Check whether the designated area exists.
Designated line is out of program range	<ul style="list-style-type: none"> • Designation was attempted of a line No. larger than the No. of lines in the entire program.

Message	Details
MDI no setting	<ul style="list-style-type: none"> Editing of the MDI was started, but the MDI setting was incomplete.
Abs/Inc mode is illegal	<ul style="list-style-type: none"> During playback editing: G90 was set when control parameter "Playback G90" was OFF. G91 was set when control parameter "Playback G90" was ON.
MDI search complete	<ul style="list-style-type: none"> The MDI search completed.
Can't search in MDI mode	<ul style="list-style-type: none"> The restart search was attempted during MDI mode. Execute the restart search after switching other than MDI mode.
MDI Set ended	<ul style="list-style-type: none"> MDI setting cannot be executed.
MDI Setting error	<ul style="list-style-type: none"> MDI setting was completed.
MDI search error	<ul style="list-style-type: none"> Could not execute the MDI search.
MDI entry complete	<ul style="list-style-type: none"> MDI entry has been completed.
Can't edit except in MDI mode	<ul style="list-style-type: none"> "#1144 mdlkof" (MDI setting lock) is "0" and therefore it is not possible to edit the MDI program in a mode other than MDI mode.
Input miss was detected	<ul style="list-style-type: none"> An input miss was detected.
Input miss was not detected	<ul style="list-style-type: none"> A search was performed for an input miss, however, none was found.
Can't edit a file except in NC memory	<ul style="list-style-type: none"> Editing cannot be performed at the edit window for programs other than those in the NC memory.
Save it	<ul style="list-style-type: none"> The cursor was tried to move beyond the editable range during mass-editing. Save, then operate again.
Save not possible	<ul style="list-style-type: none"> A special file (history file etc.) that cannot be saved was edited and an attempt was made to save it. Perform an operation to quit editing. Failed saving due to file size over, etc. during mass-editing. Setting of "#1166 fixpro" was illegal. Use the settings for a regular program. MDI cannot be entered due to MDI editing mode. Press INPUT key and finish editing.
Save left side file? (Y/N)	<ul style="list-style-type: none"> Message to confirm whether saving the left side file of the multi-program display type [Y]: Save the change [N]: Do not save the change
Save right side file? (Y/N)	<ul style="list-style-type: none"> Message to confirm whether saving the right side file of the multi-program display type [Y]: Save the change [N]: Do not save the change
Can't edit because of size over	<ul style="list-style-type: none"> The program cannot be written, because the memory capacity will be exceeded. If the memory capacity of the transfer designation device is exceeded during the automatic backup, increase the available memory of the device.
Can't edit the searched file	<ul style="list-style-type: none"> The serial file cannot be edited.
DS not ready	<ul style="list-style-type: none"> Operation was attempted when a DS was not inserted. Creating or opening a program was attempted when a DS was not inserted.
FD not ready	<ul style="list-style-type: none"> The FD operation search was attempted when an FD was not inserted. The FD list display was attempted when an FD was not inserted. Creating or opening a program was attempted when an FD was not inserted.

Message	Details
MemoryCard not ready	<ul style="list-style-type: none"> Operation was attempted when a memory card was not inserted.
This cannot be specified	<ul style="list-style-type: none"> Invalid special characters (/E, etc.) were set.
Loading	<ul style="list-style-type: none"> Loading file.
Saving	<ul style="list-style-type: none"> Saving file.
Can't execute playback edit	<ul style="list-style-type: none"> The playback editing cannot be executed because the right side area is mass-editing mode. Playback editing was attempted while program file to be edited is not designated. Display a program file in the right side area.
Program display lock C	<ul style="list-style-type: none"> The program display or search cannot be executed. Review the parameter "#1121 pglk_c" (program display lock).

6.2.11 Diagnosis-related Operation Messages

Message	Details
Erase? (Y/N)	<ul style="list-style-type: none"> Message to confirm alarm history clear operation [Y] or [INPUT] : Erase the data [N] : Do not erase the data.
Can't write data	<ul style="list-style-type: none"> The data cannot be written.
Device is illegal	<ul style="list-style-type: none"> The designated device is inadequate.
Modal output not possible	<ul style="list-style-type: none"> The modal output process failed.
Modal clear not possible	<ul style="list-style-type: none"> The modal cancel process failed.
Continue display not possible	<ul style="list-style-type: none"> Continuous display is not possible because data is not set at the cursor position.
One-shot output not possible	<ul style="list-style-type: none"> The one-shot output process failed.
Setting data not found	<ul style="list-style-type: none"> The data has not been set.
Select a menu	<ul style="list-style-type: none"> A device No. was set without selecting a menu operation. Press any operation menu and input the device No. with a menu highlighted.

6.2.12 Maintenance-related Operation Messages

Message	Details
Password is illegal	<ul style="list-style-type: none"> The input password is incorrect.
Input password	<ul style="list-style-type: none"> The [Password] menu key was pressed, and the password input mode was entered. Input a password.
Now making back-up	<ul style="list-style-type: none"> Currently backing up system data and the user PLC program to a specified device.
Backupper	<ul style="list-style-type: none"> The SRAM data is being backed up on the HD.
Backup complete	<ul style="list-style-type: none"> Backup of the SRAM data on the HD has been completed. Back up of system data and user PLC program to a specified device has been completed.
Backup error	<ul style="list-style-type: none"> An error occurred while backing up the SRAM data on the HD.
Select directory to backup	<ul style="list-style-type: none"> Select area by moving cursor, using ↑ and ↓ keys. Press the "INPUT" key to confirm.
Select directory to restore	<ul style="list-style-type: none"> Select file by moving cursor, using ↑ and ↓ keys. Press the "INPUT" key to confirm.
Executing format	<ul style="list-style-type: none"> The formatting is being executed correctly.
Format complete	<ul style="list-style-type: none"> The formatting completed correctly.
Format error	<ul style="list-style-type: none"> The NC memory formatting failed. Contact the nearest service center.
Quit HMI?(Y/N)	<ul style="list-style-type: none"> Message to confirm the HMI quitting [Y] or [INPUT] : Quit the HMI. [N] : Do not quit the HMI.
Format NC memory?(Y/N)	<ul style="list-style-type: none"> Message to confirm the NC memory formatting [Y] or [INPUT] : Start formatting the NC memory. [N] : Do not format the NC memory.
Execute SRAM backup?(Y/N)	<ul style="list-style-type: none"> Message to confirm the SRAM back up [Y] or [INPUT] : The SRAM data is backed up to the HD. [N] : The SRAM data is not backed up to the HD.
PLC running. Does it stop? (Y/N)	<ul style="list-style-type: none"> Message to confirm the PLC STOP [Y] or [INPUT] : Backup the SRAM data on the HD. [N] : Do not backup the SRAM data on the HD.
Restoring	<ul style="list-style-type: none"> The system data and user PLC program are now being restored to the NC from a specified device.
Restore complete	<ul style="list-style-type: none"> Restoration of system data and user PLC program to the NC from a specified device has been completed.
Auto adjust error	<ul style="list-style-type: none"> The H/W status cannot be read properly, and therefore it is not possible to perform auto adjustments. Check the remote I/O unit. Perform manual adjustments. The unit is defective. Replace the unit.
Change Ope. test mode	<ul style="list-style-type: none"> Operation of the Operation test screen was attempted when the operation adjustment mode was not selected.
Test mode sig valid signal is now OFF	<ul style="list-style-type: none"> The operation adjustment mode cannot be selected because the operation adjustment mode valid signal (R9998/bit0) is 0.

Message	Details
Auto adjust complete	<ul style="list-style-type: none"> • Analog output adjustment completed normally.
Auto adjust execution	<ul style="list-style-type: none"> • Performing analog output adjustment normally.
Execute? (Y/N)	<ul style="list-style-type: none"> • Message to confirm the operation [Y] or [INPUT] : The currently selected operation is performed. [N] : The currently selected operation is not performed.
Unit not equipped	<ul style="list-style-type: none"> • The machine is not equipped with an analog output unit.
A serial number is inaccurate	<ul style="list-style-type: none"> • The system data to be restored and the serial number in the NC do not match, and therefore it is not possible to restore. • Check to ensure that the serial number in the NC has been set and that the system data to be restored matches.
Operating aux axis	<ul style="list-style-type: none"> • It is not possible to set parameter and input/output data during auxiliary axis operation. • It is not possible to change the display axis during auxiliary axis operation.

6.2.13 Data Sampling-related Operation Messages

Message	Details
Executing sampling	<ul style="list-style-type: none"> The waveform data is being sampled.
Input the axis name	<ul style="list-style-type: none"> This appears when the data type is selected in the Ch1 or Ch2 data setting area. Input the name of the axis to be sampled.
Input axis name/signal No./bit	<ul style="list-style-type: none"> This appears when the data type is selected in the Ch1 or Ch2 data setting area. Set the sampling conditions common for Ch1 and Ch2.
Input device name/device No.	<ul style="list-style-type: none"> Input device name/device No.
Input file/sub-ID/item/data	<ul style="list-style-type: none"> This appears when an NC file is selected in the Ch1 or Ch2 data setting area.
Change the area	<ul style="list-style-type: none"> Change the area.
Sampling conditions are illegal	<ul style="list-style-type: none"> The data cannot be sampled as the setting conditions are illegal. Review the data, vertical scale, sampling rate and horizontal scale on the Condition setting screen.
Can't start sampling	<ul style="list-style-type: none"> "#1224 aux08/bit0" is "0" and sampling start-up cannot be performed.
The collection invalid	<ul style="list-style-type: none"> The parameters are set to prevent data being collected. Check the parameters.
The collection begin?	<ul style="list-style-type: none"> Determines whether to start data collection. Press [Y] or [INPUT] to start data collection.
The collection stop?	<ul style="list-style-type: none"> Determines whether to stop data collection. Press [Y] or [INPUT] to stop data collection.
The collection is being executed	<ul style="list-style-type: none"> An attempt was made to start data collection while data collection was being performed.
The collection is stopping	<ul style="list-style-type: none"> An attempt was made to stop data collection while data collection was stopped.
The collection invalid	<ul style="list-style-type: none"> The data collection is set invalid in the parameter. Check the parameter.
Scroll execution	<ul style="list-style-type: none"> The waveform display is being scrolled.
Refresh execution	<ul style="list-style-type: none"> The waveform is being refreshed.

6.2.14 Absolute Position Detection-related Operation Messages

Message	Details
Setting absolute position set	<ul style="list-style-type: none"> Setting from the screen was attempted of absolute position detection data when the "Absolute Position Set" was not ON. Press the menu key [Absolute Position Set] to turn it ON.
Not the abs position detection system	<ul style="list-style-type: none"> An absolute position detection system has not been selected for the currently selected axis. The machine parameter (Axis specification parameter "#2049 type") must be set.
Axis name inappropriate	<ul style="list-style-type: none"> The set axis name is inappropriate. Check the axis name.
Not passed on grid	<ul style="list-style-type: none"> The absolute position basic point was set without passing the grid after the power ON in the dogless-type absolute position detection. Return one grid back and repeat the procedure.
Can't start	<ul style="list-style-type: none"> Settings of "#0 absolute posn set", "#2 Z-point" and "#2055 pushf" are not adequate. "AbsEncoder: Serial data error" alarm (Z71 0005) has occurred. Check the parameter and the alarm.
Illegal direction	<ul style="list-style-type: none"> JOG starting direction is illegal in the machine end stopper method of the dogless-type absolute position detection.

6.2.15 System Setup-related Operation Messages

Message	Details
Initial parameter creating	<ul style="list-style-type: none"> The initial parameter is being created.
Initial parameter transferring	<ul style="list-style-type: none"> The initial parameter is being transferred.
Spindle initial parameter transferring	<ul style="list-style-type: none"> The spindle initial parameter is being transferred.
Can't write data	<ul style="list-style-type: none"> The data cannot be written. Setting of the initial parameter failed. Contact the service center.
Param set ended. Format NC memory? (Y/N)	<ul style="list-style-type: none"> After completing the parameter setting, it determines whether executing the file format or not. Enter [Y] or [INPUT] to execute the file format.
Write sample ladder? (Y/N)	<ul style="list-style-type: none"> It determines whether executing writing the sample PLC program or not. Enter [Y] or [INPUT] to start writing the sample PLC program.
Sample ladder not found	<ul style="list-style-type: none"> The file to set the sample PLC program is lost. Contact the service center.

6.2.16 Automatic Backup-related Operation Messages

Message	Details
Auto backup disabled (Device illegal)	<ul style="list-style-type: none"> Set a correct value in the device No. of the automatic backup device.
Auto backup disabled (No DS)	<ul style="list-style-type: none"> When turning ON the power next time, insert DS.
Auto backup disabled (No memory card)	<ul style="list-style-type: none"> When turning ON the power next time, insert the memory card.
Auto backup proceeding	<ul style="list-style-type: none"> Wait for the automatic backup to complete.
Auto backup completed	<ul style="list-style-type: none"> The automatic back up is completed.
Memory over	<ul style="list-style-type: none"> The memory capacity of the transfer designation device was exceeded during the automatic backup. Increase the available memory of the transfer designation device.
File access error	<ul style="list-style-type: none"> A file access error occurred during the automatic backup. Contact the service center.
Can't make directory	<ul style="list-style-type: none"> The storage destination directory of the automatic backup data described in the custom definition file does not exist. Create the directory of the storage destination.
Write protect	<ul style="list-style-type: none"> The memory card is write-protected. Release the write protection.

6.2.17 Alarm History-related Operation Messages

Message	Details
The collection begin? (Y/N)	<ul style="list-style-type: none"> Determines whether to start alarm history. Press [Y] or [INPUT] to start alarm history.
The collection stop? (Y/N)	<ul style="list-style-type: none"> Determines whether to stop alarm history. Press [Y] or [INPUT] to stop alarm history.
The collection is being executed	<ul style="list-style-type: none"> An attempt was made to start alarm history while alarm history was being performed.
The collection is stopping	<ul style="list-style-type: none"> An attempt was made to stop alarm history while alarm history was being stopped.
Execute the collection data clear?(Y/N)	<ul style="list-style-type: none"> Determines whether to clear alarm history. Press [Y] or [INPUT] to clear alarm history.
Data clear complete	<ul style="list-style-type: none"> The alarm history cleared was completed.
The collection begin	<ul style="list-style-type: none"> The alarm history was started.
The collection stop	<ul style="list-style-type: none"> The alarm history was stopped.

6.2.18 Anshin-net-related Operation Messages

(1) Messages related to all Anshin-net screens

Message		Details
(None)	<p>Press one-touch call to call the NC service.</p> <p>Do not turn the power OFF during the one-touch call.</p>	<ul style="list-style-type: none"> • Communication has not been started. • A call is being placed with automatic alarm notification or one-touch call, and a call is being received from the NC service.

(2) Messages related to automatic alarm notification

Message		Details
dialing	<p>Do not turn the power OFF until the automatic alarm notification ends.</p>	<ul style="list-style-type: none"> • A call is being placed with automatic alarm notification. • Communication starts when an alarm occurs, but the line is not connected yet in this state. • This state is also entered when standing by for transmission. • Communication has been started with emergency stop by the servo, spindle or PLC alarm, or by the establishment of the conditions set in the parameters.
Waiting for dialing	<p>Do not turn the power OFF until the automatic alarm notification ends.</p>	<ul style="list-style-type: none"> • Redialing since the NC service is using the line for other communication.
Verifying the user registration	<p>Do not turn the power OFF until the automatic alarm notification ends.</p>	<ul style="list-style-type: none"> • User authentication is being executed by the NC service side.
Connecting	<p>Do not turn the power OFF until the automatic alarm notification ends.</p>	<ul style="list-style-type: none"> • The line is connected with automatic alarm notification.
Receiving	<p>Do not turn the power OFF until the automatic alarm notification ends.</p>	<ul style="list-style-type: none"> • The diagnosis data is being received with automatic alarm notification.
Sending	<p>Do not turn the power OFF until the automatic alarm notification ends.</p>	<ul style="list-style-type: none"> • The diagnosis data is being sent with automatic alarm notification.
Transmission completed	<p>Press one-touch call to call the NC service.</p> <p>Do not turn the power OFF during the one-touch call.</p>	<ul style="list-style-type: none"> • Automatic alarm notification has ended, and the line has been disconnected.
Reception completed	<p>Press one-touch call to call the NC service.</p> <p>Do not turn the power OFF during the one-touch call.</p>	<ul style="list-style-type: none"> • Automatic alarm notification has ended, and the line has been disconnected. • This is displayed when at least one file has been received.
(Status of communication with NC service)	<p>Connecting with NC service.</p> <p>Wait for communication to end.</p>	<ul style="list-style-type: none"> • In connection standby state since line is being used by Anshin-net.

(3) Messages related to automatic alarm notification

Message		Details
Operator notice effective	If automatic operation stops while operator notification is valid, the designated telephone number will be contacted.	<ul style="list-style-type: none"> Operator notification is valid. If machining ends normally or abnormally in this state, communication with operator notification will start.
Dialing	Do not turn the power OFF until the operator notification ends.	<ul style="list-style-type: none"> Data is being transmitted with operator notification. Communication will start when machining ends normally or abnormally, but the line is not connected yet in this state. This state is also entered when standing by for transmission.
Waiting for dialing	Do not turn the power OFF until the operator notification ends.	<ul style="list-style-type: none"> Redialing since the NC service is using the line for other communication.
Verifying the user registration	Do not turn the power OFF until the operator notification ends.	<ul style="list-style-type: none"> User authentication is being executed by the NC service side.
Connecting	Do not turn the power OFF until the operator notification ends.	<ul style="list-style-type: none"> The line is connected with operator notification.
Receiving	Do not turn the power OFF until the operator notification ends.	<ul style="list-style-type: none"> The diagnosis data is being received with operator notification.
Sending	Do not turn the power OFF until the operator notification ends.	<ul style="list-style-type: none"> The diagnosis data is being sent with operator notification.
Transmission completed	Press one-touch call to call the NC service. Do not turn the power OFF during the one-touch call.	<ul style="list-style-type: none"> Operator notification has ended, and the line has been disconnected.
Reception completed	Press one-touch call to call the NC service. Do not turn the power OFF during the one-touch call.	<ul style="list-style-type: none"> Operator notification has ended, and the line has been disconnected. This is displayed when at least one file has been received.
(Status of communication with NC service)	Connecting with NC service. Wait for communication to end.	<ul style="list-style-type: none"> In connection standby state since line is being used by Anshin-net.

(4) Messages related to automatic alarm notification

Message		Details
Carry out one-touch call? (Y/N)	Press "Y" to make a one-touch call and "N" to cancel. If the line is being in use, a connection with NC service will be established as soon as the line becomes idle.	<ul style="list-style-type: none"> • A connection with NC service has not been established. • The system is confirming whether to actually make a one-touch call. • This is displayed when the Call menu is pressed. Press Y or INPUT to execute one-touch call.

Message		Details
Dialing	Do not turn the power OFF until the one-touch call ends.	<ul style="list-style-type: none"> • Data is being transmitted with one-touch call. • Communication will start when one-touch call is executed, but the line is not connected yet in this state. • This state is also entered when standing by for transmission.
Waiting for dialing	Do not turn the power OFF until the one-touch call ends.	<ul style="list-style-type: none"> • Redialing since the NC service is using the line for other communication.
Verifying the user registration	Do not turn the power OFF until the one-touch call ends.	<ul style="list-style-type: none"> • User authentication is being executed by the NC service side.
Connecting	Do not turn the power OFF until the one-touch call ends.	<ul style="list-style-type: none"> • The line is connected with one-touch call.
Receiving	Do not turn the power OFF until the one-touch call ends.	<ul style="list-style-type: none"> • The diagnosis data is being received with one-touch call.
Sending	Do not turn the power OFF until the one-touch call ends.	<ul style="list-style-type: none"> • The diagnosis data is being sent with one-touch call.
Transmission completed	Press one-touch call to call the NC service. Do not turn the power OFF during the one-touch call.	<ul style="list-style-type: none"> • Communication with one-touch call has ended, and the line has been disconnected.
Reception completed	Press one-touch call to call the NC service. Do not turn the power OFF during the one-touch call.	<ul style="list-style-type: none"> • Communication with one-touch call has ended, and the line has been disconnected. • This is displayed when at least one file has been received.
(Status of communication with NC service)	Connecting with NC service. Wait for communication to end.	<ul style="list-style-type: none"> • In connection standby state since line is being used by Anshin-net.

(5) Messages related to transmission request from NC service

Message		Details
Verifying the destination	Connecting with NC service. Wait for communication to end.	<ul style="list-style-type: none"> User authentication is being executed by the NC system side.
Connecting	Connecting with NC service. Wait for communication to end.	<ul style="list-style-type: none"> The line is connected upon transmission request from NC service.
Receiving	Connecting with NC service. Wait for communication to end.	<ul style="list-style-type: none"> Data is being received upon transmission request from NC service.
Sending	Connecting with NC service. Wait for communication to end.	<ul style="list-style-type: none"> Data is being sent upon transmission request from NC service.
Transmission completed	Press one-touch call to call the NC service. Do not turn the power OFF during the one-touch call.	<ul style="list-style-type: none"> Transmission request from NC service has been completed, and the line has been disconnected.
Reception completed	Press one-touch call to call the NC service. Do not turn the power OFF during the one-touch call.	<ul style="list-style-type: none"> Transmission request from NC service has been completed, and the line has been disconnected. This is displayed when at least one file or message has been received.

(6) Messages related to number 1 to 3 menu operations

Message		Details
(None)	Press one-touch call to call the NC service. Do not turn the power OFF during the one-touch call.	<ul style="list-style-type: none"> The selected telephone No. will be set as the telephone No. to be notified to the NC service. Hereafter, the telephone No. set with one-touch call or operator notification will be notified to the NC service.

(7) Messages related to arbitrary number setting

Message		Details
(None)	Input the telephone No. to be notified.	<ul style="list-style-type: none"> The input telephone No. will be set as the telephone No. to be notified to the NC service. Hereafter, the telephone No. set with one-touch call or operator notification will be notified to the NC service.

(8) Messages related to sharing of machining data

Message	Details
Transmit by the set password?(Y/N)	<ul style="list-style-type: none"> The line is not connected with NC service. The system is confirming whether to transfer machining data. Press "Y" or "INPUT" to transfer machining data, "N" to cancel.
OK? (Y/N)	<ul style="list-style-type: none"> The line is not connected with NC service. The system is confirming whether to erase machining data. Press "Y" or "INPUT" to erase machining data, "N" to cancel.
dialing	<ul style="list-style-type: none"> A call is being placed. "Transmit by the set password?(Y/N)" or "OK? (Y/N)" is shown. Pressing "Y" or "INPUT" starts the communication. The line is not connected yet in this state.
Connecting	<ul style="list-style-type: none"> The line is connected for sharing of machining data.
Transmitting	<ul style="list-style-type: none"> The machining data is being transmitted.
Transmission completed	<ul style="list-style-type: none"> Machining data transmission has ended, and the line has been disconnected.
Receiving	<ul style="list-style-type: none"> The machining data is being received.
Reception completed	<ul style="list-style-type: none"> Machining data reception has ended, and the line has been disconnected.
Erase complete	<ul style="list-style-type: none"> Machining data erasing has ended, and the line has been disconnected.
Waiting for dialing	<ul style="list-style-type: none"> In dialing standby state since the line is being used.
Input the password	<ul style="list-style-type: none"> The password, which is required for the transmission/reception of machining data, has not been set. Input the password set on the Anshin-net parameter 1 screen.
Input the user number	<ul style="list-style-type: none"> The user number, which is required for the transmission/reception of machining data, has not been set. Input the user number set on the Anshin-net parameter 1 screen.

6.2.19 Messages Related to Machine Tool Builder Network System

(1) Messages related to all Machine Tool Builder Network System (MTB net) Screens.

Message		Details
(None)	Press the [Send] menu when you transmit the diagnosis data to MTB. Don't turn OFF the power supply while transmitting.	<ul style="list-style-type: none"> Communication has not been started.

Message	Details
Network service is connected	<ul style="list-style-type: none"> The settings of the MTB net parameter 1,2 cannot be changed since the system is communicating with Call Center or machine tool builder. Set again after the communication has ended.

(2) Messages related to transmission of diagnosis data

Message		Details
Transmit diagnosis data? (Y/N)	Press Y (transmit the diagnosis data) or N (cancel). You'll connect with MTB when it gets available if line used.	<ul style="list-style-type: none"> The system is confirming the transmission of diagnosis data.
dialing	Don't turn OFF a power supply until diagnosis data transmission ends.	<ul style="list-style-type: none"> The diagnosis data is being transmitted. The line is not connected yet in this state.
Verifying the user registration	Don't turn OFF a power supply until diagnosis data transmission ends.	<ul style="list-style-type: none"> The system is waiting for an authentication response from remote diagnosis tool kit.
Waiting for the reply	Don't turn OFF a power supply until diagnosis data transmission ends.	<ul style="list-style-type: none"> The line has been disconnected once, and the system is waiting for a connection with the machine tool builder.
Verifying the destination	Don't turn OFF a power supply until diagnosis data transmission ends.	<ul style="list-style-type: none"> The line has been connected corresponding to the received call from the machine tool builder. The system is confirming the destination of connection.
Connecting	Don't turn OFF a power supply until diagnosis data transmission ends.	<ul style="list-style-type: none"> The system is connected or connecting with machine tool builder.
Transmitting	Don't turn OFF a power supply until diagnosis data transmission ends.	<ul style="list-style-type: none"> The diagnosis data is actually being transmitted.
Transmission completed	Press Y (transmit the diagnosis data) or N (cancel). You'll connect with MTB when it gets available if line used.	<ul style="list-style-type: none"> The transmission of diagnosis data has ended, and the line has been disconnected.

Message		Details
Waiting for dialing	Don't turn OFF a power supply until diagnosis data transmission ends.	<ul style="list-style-type: none"> The machine tool builder is using the line for other communication.
(Status of communication with NC service)	Connecting with NC service. Please wait until the communication ends.	<ul style="list-style-type: none"> In connection standby state since line is being used by Anshin-net.
(Status of communication with machine manufacturer)	Connecting with MTB. Please wait until the communication ends.	<ul style="list-style-type: none"> In connection standby state since line is being used by the machine tool builder.

(3) Messages related to reception of the diagnosis results

Message		Details
Verifying the destination	Connecting with MTB. Please wait until the communication ends.	<ul style="list-style-type: none"> The line has been connected corresponding to the received call from the machine tool builder. The system is confirming the destination of connection.
Connecting	Connecting with MTB. Please wait until the communication ends.	<ul style="list-style-type: none"> The system is connected or connecting with machine tool builder.
Receiving	Connecting with MTB. Please wait until the communication ends.	<ul style="list-style-type: none"> The diagnosis results are actually being received.
Reception completed	Press the [Send] menu when you transmit the diagnosis data to MTB. Don't turn OFF the power supply while transmitting.	<ul style="list-style-type: none"> Reception of the diagnosis results has ended, and the line has been disconnected.

(4) Messages related to reception of messages

Message		Details
Verifying the destination	Connecting with MTB. Please wait until the communication ends.	<ul style="list-style-type: none"> The line has been connected corresponding to the received call from the machine tool builder. The system is confirming the destination of connection.
Connecting	Connecting with MTB. Please wait until the communication ends.	<ul style="list-style-type: none"> The system is connected or connecting with machine tool builder.
Reception completed	Press the [Send] menu when you transmit the diagnosis data to MTB. Don't turn OFF the power supply while transmitting.	<ul style="list-style-type: none"> Reception of the message has ended, and the line has been disconnected.

6.2.20 Other Operation Messages

Message	Details
Executing automatic operation	<ul style="list-style-type: none"> Cannot be performed during automatic operation. Perform the operation again after automatic operation has been completed.
Setting error	<ul style="list-style-type: none"> The setting data is inadequate. (Alphabetic characters were set where only numeric characters can be set, etc.) The data has not been set. There is no specification.
Data range error	<ul style="list-style-type: none"> The input data exceeded the range. Set the value again within the range.
Data protect	<ul style="list-style-type: none"> Setting, erasing, parameter setting, etc., of the various data is prohibited, because the data protect key is validated. Reconsider the data protect key setting.
Write protect	<ul style="list-style-type: none"> Attempted to create a new program file in the write-protected device Opened the program file of the write-protected device Attempted to save the program file of the write-protected device Attempted to correct the buffer for the write-protected file. Attempted to edit or correct the buffer for the read-only program file.
Edit lock B	<ul style="list-style-type: none"> It is not possible to change machining program B (8000 to 8999: user standard subprogram) or machining program C (9000 to 9999: machine tool builder custom program) as edit lock B is enabled.
Edit lock C	<ul style="list-style-type: none"> It is not possible to change machining program C (9000 to 9999: machine tool builder custom program) as edit lock C is enabled.
Origin set not possible	<ul style="list-style-type: none"> The operation is in a state in which origin set is not possible. Check the parameter "#1123 origin (Origin set prohibited)" setting. Check that the axis has stopped. Check that idling (post-reset status) is currently being performed.
Can't command manual value	<ul style="list-style-type: none"> The manual numerical value protect is valid and therefore it is not possible to perform a manual numerical value command.
Getting T code list	<ul style="list-style-type: none"> T code list is being retrieved.
T code list complete	<ul style="list-style-type: none"> Retrieving T code list is completed.
Load meter display not possible	<ul style="list-style-type: none"> The load meter cannot be displayed. Contact the machine tool builder.
Pallet running	<ul style="list-style-type: none"> Each setting was executed during pallet running.
APC executing	<ul style="list-style-type: none"> Each setting was executed during automatic pallet changer executing.

6.3 Program Error

(The bold characters are the message displayed in the screen.)

These alarms occur during automatic operation, and the causes of these alarms are mainly program errors which occur, for instance, when mistakes have been made in the preparation of the machining programs or when programs which conform to the specification have not been prepared.

Error No.	Details	Remedy
P 10	No. of simultaneous axes over The number of axis addresses commanded in the same block exceeds the specifications.	<ul style="list-style-type: none"> • Divide the alarm block command into two. • Check the specifications.
P 11	Illegal axis address The axis address commanded by the program and the axis address set by the parameter do not match.	<ul style="list-style-type: none"> • Revise the axis names in the program.
P 20	Division error An axis command which cannot be divided by the command unit has been issued.	<ul style="list-style-type: none"> • Check the program.
P 29	Not accept command The normal line control command (G40.1, G41.1, G42.1) has been issued during the modal in which the normal line control is not acceptable.	<ul style="list-style-type: none"> • Check the program.
P 30	Parity H error The number of holes per character on the paper tape is even for EIA code and odd for ISO code.	<ul style="list-style-type: none"> • Check the paper tape. • Check the tape puncher and tape reader.
P 31	Parity V error The number of characters per block on the paper tape is odd.	<ul style="list-style-type: none"> • Make the number of characters per block on the paper tape even. • Set the parameter parity V selection OFF.
P 32	Illegal address An address not listed in the specifications has been used.	<ul style="list-style-type: none"> • Check and revise the program address. • Check and correct the parameters values. • Check the specifications.
P 33	Format error The command format in the program is not correct.	<ul style="list-style-type: none"> • Check the program.
P 34	Illegal G code A G code not listed in the specifications has been used. An illegal G code was commanded during the coordinate rotation command (G68).	<ul style="list-style-type: none"> • Check and correct the G code address in the program.
	G51.2 or G50.2 was commanded when the rotary tool axis No. (#1501 polyax) was set to "0". G51.2 or G50.2 was commanded when the tool axis was set to the linear axis (#1017 rot "0").	<ul style="list-style-type: none"> • Check the parameter setting values.
P 35	Setting value range over The setting range for the addresses has been exceeded.	<ul style="list-style-type: none"> • Check the program.
P 36	Program end error "EOR" has been read during tape and memory mode.	<ul style="list-style-type: none"> • Enter the M02 and M30 command at the end of the program. • Enter the M99 command at the end of the subprogram.

Error No.	Details	Remedy
P 37	O, N number zero A zero has been specified for program and sequence Nos.	<ul style="list-style-type: none"> The program Nos. are designated across a range from 1 to 99999999. The sequence Nos. are designated across a range from 1 to 99999.
P 38	No spec: Add. Op block skip "/n" has been issued even though there are no optional block skip addition specifications.	<ul style="list-style-type: none"> Check the specifications.
P 39	No specifications <ul style="list-style-type: none"> A non-specified G code was specified. The selected operation mode is not used. 	<ul style="list-style-type: none"> Check the specifications.
P 40	Pre-read block error When tool radius compensation is executed, there is an error in the pre-read block and so the interference check is disabled.	<ul style="list-style-type: none"> Reconsider the program.
P 48	Restart pos return incomplete Movement command was executed before executing the block that is restart-searched.	<ul style="list-style-type: none"> Carry out program restart again. Movement command cannot be executed before executing the block that is restart-searched.
P 49	Invalid restart search <ul style="list-style-type: none"> Restart search was attempted for the 3-dimensional circular interpolation. Restart search was attempted during the cylindrical interpolation, polar coordinate interpolation, and tool tip center control. 	<ul style="list-style-type: none"> Reconsider the program. Reconsider the restart search position.
P 50	No spec: Inch/Metric change Inch/Metric changeover (G20/G21) command was issued even though there is no inch/metric conversion specification.	<ul style="list-style-type: none"> Check the specifications.
P 60	Compensation length over The commanded movement distance is excessive. (Over 2 ³¹)	<ul style="list-style-type: none"> Reconsider the axis address command.
P 61	No spec: Unidirectional posit. Unidirectional positioning (G60) was commanded even though there is no unidirectional positioning specification.	<ul style="list-style-type: none"> Check the specifications.
P 62	No F command <ul style="list-style-type: none"> No feed rate command has been issued. There is no F command in the cylindrical interpolation or polar coordinate interpolation immediately after the G95 mode is commanded. 	<ul style="list-style-type: none"> The default movement modal command at power ON is G01. This causes the machine to move without a G01 command if a movement command is issued in the program, and an alarm results. Use an F command to specify the feed rate. Specify F with a thread lead command.
P 63	No spec: High-speed machining High-speed machining cancel (G5P0) was commanded even though there is no high-speed machining mode specification.	<ul style="list-style-type: none"> Check the specifications.
P 65	No spec: High speed mode 3	<ul style="list-style-type: none"> Check the high-speed mode III specifications.

Error No.	Details	Remedy
P 70	Arc end point deviation large <ul style="list-style-type: none"> • There is an error in the arc start and end points as well as in the arc center. • The difference of the involute curve through the start point and the end point is large. • When arc was commanded, one of the two axes configuring the arc plane was a scaling valid axis. 	<ul style="list-style-type: none"> • Check the numerical values of the addresses that specify the start and end points, arc center as well as the radius in the program. • Check the "+" and "-" directions of the address numerical values. • Check the scaling valid axis.
P 71	Arc center error <ul style="list-style-type: none"> • The arc center is not sought during R-specified circular interpolation. • The curvature center of the involute curve cannot be obtained. 	<ul style="list-style-type: none"> • Check the numerical values of the addresses in the program. • Check whether the start point or end point is on the inner side of the base circle for involute interpolation. When carrying out tool radius compensation, check that the start point and end point after compensation are not on the inner side of the base circle for involute interpolation. • Check whether the start point and end point are at an even distance from the center of the base circle for involute interpolation.
P 72	No spec: Helical cutting A helical command has been issued though it is not included in the specifications.	<ul style="list-style-type: none"> • Check the helical specifications. • An Axis 3 command was issued by the circular interpolation command. If there is no helical specification, the linear axis is moved to the next block.
P 73	No spec: Spiral cutting A spiral command was issued despite the fact that such a command does not exist in the specifications.	<ul style="list-style-type: none"> • The G02.1 and G03.1 commands are issued for circular interpolation. • Check the spiral specifications.
P 74	Can't calculate 3DIM arc The end block was not specified during 3-dimension circular interpolation supplementary modal, and therefore it is not possible to calculate the 3-dimension circular interpolation. Furthermore, it not possible to calculate the 3-dimension circular interpolation due to an interruption during 3-dimension circular interpolation supplementary modal.	<ul style="list-style-type: none"> • Reconsider the program.
P 75	3DIM arc illegal An unusable G code was issued during 3-dimension circular interpolation modal. Or, a 3-dimension circular interpolation command was issued during a modal for which a 3-dimension circular interpolation command cannot be issued.	<ul style="list-style-type: none"> • Reconsider the program.
P 76	No spec: 3DIM arc interpolat G02.4/G03.4 was commanded even though there is no 3-dimension circular interpolation specification.	<ul style="list-style-type: none"> • Check the specifications.
P80	No spec: Hypoth ax interpolat Hypothetical axis interpolation (G07) was commanded even though there is no hypothetical axis interpolation specification.	<ul style="list-style-type: none"> • Check the specifications.

Error No.	Details	Remedy
P 90	No spec: Thread cutting A thread cutting command was issued even though there is no thread cutting command specification.	<ul style="list-style-type: none"> • Check the specifications.
P 91	No spec: Var lead threading Variable lead thread cutting (G34) was commanded even though there is no variable lead thread cutting specification.	<ul style="list-style-type: none"> • Check the specifications.
P 93	Illegal pitch vaule The thread lead (thread pitch) when performing the thread cutting command is incorrect.	<ul style="list-style-type: none"> • Set the correct thread lead command for the thread cutting command.
P100	No spec: Cylindric interpolat A cylindrical interpolation command was issued even though there is no cylindrical interpolation specification.	<ul style="list-style-type: none"> • Check the specifications.
P110	Plane select during figure rot Plane selection (G17/G18/G19) was commanded during figure rotation.	<ul style="list-style-type: none"> • Check the machining program.
P111	Plane selected while coord rot Plane selection commands (G17, G18, G19) were issued during a coordinate rotation command (G68).	<ul style="list-style-type: none"> • After command G68, always issue a plane selection command following a G69 (coordinate rotation cancel) command.
P112	Plane selected while R compen <ul style="list-style-type: none"> • Plane selection commands (G17, G18, G19) were issued while tool radius compensation (G41, G42) and nose R compensation (G41, G42, G46) commands were being issued. • Plane selection commands were issued after completing nose R compensation commands when there are no further axis movement commands after G40, and compensation has not been cancelled. 	<ul style="list-style-type: none"> • Issue plane selection commands after completing (axis movement commands issued after G40 cancel command) tool radius compensation and nose R compensation commands.
P113	Illegal plane select The circular command axis differs from the selected plane.	<ul style="list-style-type: none"> • Issue a circular command after correct plane selection.
P120	No spec: Feed per rotation Feed per rotation (G95) was commanded even though there is no feed per rotation specification.	<ul style="list-style-type: none"> • Check the specifications.
P121	F0 command during arc modal F0 (F 1-digit feed) was commanded during the arc modal (G02/G03).	<ul style="list-style-type: none"> • Check the machining program.
P122	No spec: Auto corner override An auto corner override command (G62) was issued even though there is no auto corner override specification.	<ul style="list-style-type: none"> • Check the specifications. • Delete the G62 command from the program.
P123	No spec: High-accuracy control High-accuracy control command was issued even though there is no high-accuracy control specification	<ul style="list-style-type: none"> • Check the specifications.

Error No.	Details	Remedy
P124	No spec: Inverse time feed There is no inverse time option.	<ul style="list-style-type: none"> • Check the specifications.
P125	G93 mode error <ul style="list-style-type: none"> • A G code command that cannot be issued was issued during G93 mode. • G93 command was issued during a modal for which inverse time feed cannot be performed. 	<ul style="list-style-type: none"> • Reconsider the program.
P126	Invalid cmd in high-accuracy An illegal command was issued during the high-accuracy control mode. <ul style="list-style-type: none"> • A G code group 13 command was issued during the high-accuracy control mode. • Milling, cylindrical interpolation or pole coordinate interpolation was commanded during the high-accuracy control mode. 	<ul style="list-style-type: none"> • Reconsider the program.
P127	No spec: SSS Control The SSS control valid parameter was set to ON although there is no SSS control specification.	<ul style="list-style-type: none"> • Check the specifications. If there is no SSS control specification, set the parameter #8090 SSS ON to 0.
P130	2nd M function code illegal The 2nd miscellaneous function address commanded in the program differs from the address set in the parameters. miscellaneous function.	<ul style="list-style-type: none"> • Check and correct the 2nd miscellaneous function address in the program.
P131	No spec: Cnst surface ctrl G96 A constant surface speed control command (G96) was issued even though there is no specification.	<ul style="list-style-type: none"> • Check the specifications. • Change the constant surface speed control command (G96) to a rotation speed command (G97).
P132	Spindle rotation speed S=0 No spindle rotation speed command has been issued.	<ul style="list-style-type: none"> • Reconsider the program.
P133	Illegal P-No. G96 An invalid constant surface speed control axis has been specified.	<ul style="list-style-type: none"> • Reconsider the parameter specified for the constant surface speed control axis.
P140	No spec: Pos compen cmd The position compensation command (G45 to G48) specifications are not available.	<ul style="list-style-type: none"> • Check the specifications.
P141	Pos compen during rotation Position compensation was commanded during the figure rotation or coordinate rotation command.	<ul style="list-style-type: none"> • Reconsider the program.
P142	Pos compen invalid arc A position compensation invalid arc command was commanded.	<ul style="list-style-type: none"> • Reconsider the program.

Error No.	Details	Remedy
P150	No spec: Nose R compensation <ul style="list-style-type: none"> • Even though there were no tool radius compensation specifications, tool radius compensation commands (G41 and G42) were issued. • Even though there were no nose R compensation specifications, nose R compensation commands (G41, G42, and G46) were issued. 	<ul style="list-style-type: none"> • Check the specifications.
P151	Radius compen during arc mode A compensation command (G40, G41, G42, G43, G44, G46) has been issued in the arc modal (G02, G03).	<ul style="list-style-type: none"> • Issue the linear command (G01) or rapid traverse command (G00) in the compensation command block or cancel block. (Set the modal to linear interpolation.)
P152	No intersection In interference block processing during execution of a tool radius compensation (G41 or G42) or nose R compensation (G41, G42, or G46) command, the intersection point after one block is skipped cannot be determined.	<ul style="list-style-type: none"> • Reconsider the program.
P153	Compensation interference An interference error has arisen while the tool radius compensation command (G41, G42) or nose R compensation command (G41, G42, G46) was being executed.	<ul style="list-style-type: none"> • Reconsider the program.
P154	No spec: 3D compensation A three-dimensional compensation command was issued even though there are no three-dimensional compensation specifications.	<ul style="list-style-type: none"> • Check the specifications.
P155	Fixed cyc exec during compen A fixed cycle command has been issued in the radius compensation mode.	<ul style="list-style-type: none"> • The radius compensation mode is established when a fixed cycle command is executed and so the radius compensation cancel command (G40) should be issued.
P156	R compen direction not defined At the start of G46 nose R compensation, the compensation direction is undefined if this shift vector is used.	<ul style="list-style-type: none"> • Change the vector to that with which the compensation direction is defined. • Exchange with a tool having a different tip point No.
P157	R compen direction changed During G46 nose R compensation, the compensation direction is inverted.	<ul style="list-style-type: none"> • Change the G command to that which allows inversion of the compensation direction (G00, G28, G30, G33, or G53). • Exchange with a tool having a different tip point No. • Turn ON the "#8106 G46 NO REV-ERR" parameter.
P158	Illegal tip point During G46 nose R compensation, the tip point is illegal (other than 1 to 8).	<ul style="list-style-type: none"> • Change the tip point No. to a legal one.

Error No.	Details	Remedy
P170	No offset number The compensation No. (DOO, TOO, HOO) command was not given when the radius compensation (G41, G42, G43, G46) command was issued. Alternatively, the compensation No. is larger than the number of sets in the specifications.	<ul style="list-style-type: none"> • Add the compensation No. command to the compensation command block. • Check the number of compensation No. sets a correct it to a compensation No. command within the permitted number of tool compensation sets.
P171	No spec:Comp input by prog G10 Compensation data input by program (G10) was commanded even though there is no specification of compensation data input by program.	<ul style="list-style-type: none"> • Check the specifications.
P172	G10 L number error (G10 L-No. error) The L address command is not correct when the G10 command is issued.	<ul style="list-style-type: none"> • Check the address L-No. of the G10 command and correct the No.
P173	G10 P number error (G10 compensation error) When the G10 command is issued, a compensation No. outside the permitted number of sets in the specifications has been commanded for the compensation No. command.	<ul style="list-style-type: none"> • First check the number of compensation sets and then set the address P designation to within the permitted number of sets.
P174	No spec:Comp input by prog G11 Compensation data input by program cancel (G11) was commanded even though there is no specification of compensation data input by program.	<ul style="list-style-type: none"> • Check the specifications.
P177	Tool life count active Registration of tool life management data with G10 was attempted when the used data count valid signal was ON.	<ul style="list-style-type: none"> • The tool life management data cannot be registered when counting the used data. Turn the used data count valid signal OFF.
P178	Tool life data entry over The number of registration groups, total number of registered tools or the number of registrations per group exceeded the specifications range.	<ul style="list-style-type: none"> • Review the number of registrations.
P179	Illegal group No. <ul style="list-style-type: none"> • When registering the tool life management data with G10, the group No. was commanded in duplicate. • A group No. that was not registered was designated during the T□□□□99 command. • An M code command must be issued as a single command but coexists in the same block as that of another M code command. • The M code commands set in the same group exist in the same block. 	<ul style="list-style-type: none"> • The group No. cannot be commanded in duplicate. When registering the group data, register it in group units. • Correct to the correct group No.
P180	No spec: Drilling cycle A fixed cycle command was issued though there are not fixed cycle (G72 - G89) specifications.	<ul style="list-style-type: none"> • Check the specifications. • Correct the program.

Error No.	Details	Remedy
P181	No spindle command (Tap cycle) The spindle rotation speed command has not been issued when the fixed cycle for drilling command is given. "S*****" type S command does not exist in the same block with the synchronous tapping cycle.	<ul style="list-style-type: none"> Issue the spindle rotation speed command (S) when the fixed cycle for drilling command G84, G74 (G84, G88) is given. Enter "S*****" type S command.
P182	Synchronous tap error <ul style="list-style-type: none"> Connection to the main spindle unit was not established. The synchronous tapping was attempted with the spindle not serially connected under the multiple-spindle control I. 	<ul style="list-style-type: none"> Check connection to the main spindle. Check that the main spindle encoder exists. Set 1 to the parameter #3024 (sout).
P183	No pitch/thread number The pitch or thread number command has not been issued in the tap cycle of a fixed cycle for drilling command.	<ul style="list-style-type: none"> Specify the pitch data and the number of threads by F or E command.
P184	Pitch/thread number error <ul style="list-style-type: none"> The pitch or the number of threads per inch is illegal in the tap cycle of the fixed cycle for drilling command. The pitch is too small for the spindle rotation speed. The thread number is too large for the spindle rotation speed. 	<ul style="list-style-type: none"> Check the pitch or the number of threads per inch.
P185	No spec: Sync tapping cycle Synchronous tapping cycle (G84/G74) was commanded even though there is no synchronous tapping cycle specification.	<ul style="list-style-type: none"> Check the specifications.
P186	Illegal S cmdnd in synchro tap S command was issued during synchronous tapping modal.	<ul style="list-style-type: none"> Cancel the synchronous tapping before issuing the S command.
P190	No spec: Turning cycle A lathe cutting cycle command was input although the lathe cutting cycle was undefined in the specification.	<ul style="list-style-type: none"> Check the specification. Delete the lathe cutting cycle command.
P191	Taper length error In the lathe cutting cycle, the specified length of taper section is illegal.	<ul style="list-style-type: none"> The radius set value in the lathe cycle command must be smaller than the axis shift amount.
P192	Chamfering error Chamfering in the thread cutting cycle is illegal.	<ul style="list-style-type: none"> Set a chamfering amount not exceeding the cycle.
P200	No spec: MRC cycle The compound type fixed cycle for turning machining I (G70 to G73) was commanded when the compound type fixed cycle for turning machining I specifications were not provided.	<ul style="list-style-type: none"> Check the specification.

Error No.	Details	Remedy
P201	Program error (MRC) <ul style="list-style-type: none"> • When called with a compound type fixed cycle for turning machining I command, the subprogram contained at least one of the following commands: <ul style="list-style-type: none"> • Reference position return command (G27, G28, G29, G30) • Thread cutting (G33, G34) • Fixed cycle skip-function (G31, G31.n) • The first move block of the finish shape program in compound type fixed cycle for turning machining I contains an arc command. 	<ul style="list-style-type: none"> • Delete the following G codes from this subprogram that is called with the compound type fixed cycle for turning machining I commands (G70 to G73): G27, G28, G29, G30, G31, G33, G34, and fixed cycle G codes. • Remove G2 and G3 from the first move block of the finish shape program in compound type fixed cycle for turning machining I.
P202	Block over (MRC) The number of blocks in the shape program of the compound type fixed cycle for turning machining I is over 50 or 200 (this differs according to the model).	<ul style="list-style-type: none"> • Specify 50 or a less value. The number of blocks in the shape program called by the compound type fixed cycle for turning machining I commands (G70 to G73) must be decreased below 50 or 200 (this differs according to the model).
P203	D cmd figure error (MRC) The compound type fixed cycle for turning machining I (G70 to G73) shape program could not cut the work normally because it defined an abnormal shape.	<ul style="list-style-type: none"> • Check the compound type fixed cycle for turning machining I (G70 to G73) shape program.
P204	E cmd fixed cycle error A command value of the compound type fixed cycle for turning machining (G70 to G76) is illegal.	<ul style="list-style-type: none"> • Check the compound type fixed cycle for turning machining (G70 to G76) command value.
P210	No spec: Pattern cycle A compound type fixed cycle for turning machining II (G74 to G76) command was input although it was undefined in the specification.	<ul style="list-style-type: none"> • Check the specification.
P220	No spec: Special fixed cycle No special fixed cycle specifications are available.	<ul style="list-style-type: none"> • Check the specifications.
P221	No. of special fixed holes = 0 A 0 has been specified for the number of holes in special fixed cycle mode.	<ul style="list-style-type: none"> • Reconsider the program.
P222	G36 angle error A G36 command specifies 0 for angle intervals.	<ul style="list-style-type: none"> • Reconsider the program.
P223	G12/G13 radius error The radius value specified with a G12 or G13 command is below the compensation amount.	<ul style="list-style-type: none"> • Reconsider the program.
P224	No spec: Circular (G12/G13) There are no circular cutting specifications.	<ul style="list-style-type: none"> • Check the specifications.

Error No.	Details	Remedy
P230	Subprogram nesting over <ul style="list-style-type: none"> A subprogram has been called 8 or more times in succession from the subprogram. The program in the data server contains the M198 command. The program in the IC card has been called more than once (the program in the IC card can be called only once at a time). 	<ul style="list-style-type: none"> Check the number of subprogram calls and correct the program so that it does not exceed 8 times.
P231	No sequence No. At subprogram call time, the sequence No. set at return from the subprogram or specified by GOTO, was not set.	<ul style="list-style-type: none"> Specify the sequence Nos. in the call block of the subprogram.
P232	No program No. <ul style="list-style-type: none"> The machining program has not been found when the machining program is called. The file name of the program registered in IC card is not corresponding to O No. 	<ul style="list-style-type: none"> Enter the machining program. Check the subprogram storage destination parameters. Ensure that the external device (including IC card) that contains the file is mounted.
P235	Program editing Operation was attempted for the file under program editing.	<ul style="list-style-type: none"> Execute the program again after completion of program editing.
P240	Program editing Operation was attempted for the file under program editing.	<ul style="list-style-type: none"> Check the specifications.
P241	No variable No. The variable No. commanded is out of the range specified in the specifications.	<ul style="list-style-type: none"> Check the specifications. Check the program variable No.
P242	= not defined at vrble set The "=" sign has not been commanded when a variable is defined.	<ul style="list-style-type: none"> Designate the "=" sign in the variable definition of the program.
P243	Can't use variables An invalid variable has been specified in the left or right side of an operation expression.	<ul style="list-style-type: none"> Correct the program.
P244	Invalid set date or time Date or time was set earlier than current date or time in the system variables (#3011, #3012) when the credit system was valid.	<ul style="list-style-type: none"> Date or time cannot be changed. Reconsider the program.
P250	No spec: Figure rotation Figure rotation (M98 I_J_P_H_L_) was commanded even though there is no figure rotation specification.	<ul style="list-style-type: none"> Check the specifications.
P251	Figure rotation overlapped Figure rotation command was issued during figure rotation.	<ul style="list-style-type: none"> Check the machining program.
P252	Coord rotate in fig. rotation A coordinate rotation related command (G68, G69) was issued during figure rotation.	<ul style="list-style-type: none"> Reconsider the program.
P260	No spec: Coordinates rotation Even though there were no coordinate rotation specifications, a coordinate rotation command was issued.	<ul style="list-style-type: none"> Check the specifications.

Error No.	Details	Remedy
P270	No spec: User macro A macro specification was commanded though there are no such command specifications.	<ul style="list-style-type: none"> Check the specifications.
P271	No spec: Macro interrupt A macro interruption command has been issued though it is not included in the specifications.	<ul style="list-style-type: none"> Check the specifications.
P272	NC and macro texts in a block A statement and a macro statement exist together in the same block.	<ul style="list-style-type: none"> Reconsider the program and place the executable statement and macro statement in separate blocks.
P273	Macro call nesting over The number of macro call nests exceeded the specifications.	<ul style="list-style-type: none"> Reconsider the program and correct it so that the macro calls do not exceed the limit imposed by the specification.
P275	Macro argument over The number of macro call argument type II sets has exceeded the limit.	<ul style="list-style-type: none"> Reconsider the program.
P276	Illegal G67 command A G67 command was issued though it was not during the G66 command modal.	<ul style="list-style-type: none"> Reconsider the program. The G67 command is the call cancel command and so the G66 command must be designated first before it is issued.
P277	Macro alarm message An alarm command has been issued in #3000.	<ul style="list-style-type: none"> Refer to the operator messages on the DIAG screen. Refer to the instruction manual issued by the machine tool builder.
P280	Brackets [] nesting over The number of parentheses "[" or "]" which can be commanded in a single block has exceeded five.	<ul style="list-style-type: none"> Reconsider the program and correct it so the number of "[" or "]" is five or less.
P281	Brackets [] not paired The number of "[" and "]" parentheses commanded in a single block does not match.	<ul style="list-style-type: none"> Reconsider the program and correct it so that "[" and "]" parentheses are paired up properly.
P282	Calculation impossible The arithmetic formula is incorrect.	<ul style="list-style-type: none"> Reconsider the program and correct the formula.
P283	Divided by zero The denominator of the division is zero.	<ul style="list-style-type: none"> Reconsider the program and correct it so that the denominator for division in the formula is not zero.
P290	IF sentence error There is an error in the IF conditional GOTO□ statement.	<ul style="list-style-type: none"> Reconsider the program.
P291	WHILE sentence error There is an error in the WHILE conditional DO□-END□ statement.	<ul style="list-style-type: none"> Reconsider the program.
P292	SETVN sentence error There is an error in the SETVN□ statement when the variable name setting was made.	<ul style="list-style-type: none"> Reconsider the program. The number of characters in the variable name of the SETVN statement must be 7 or less.
P293	DO-END nesting over The number of DO-END nesting levels in WHILE conditional DO□-END□ statement has exceeded 27.	<ul style="list-style-type: none"> Reconsider the program and correct it so that the nesting levels of the DO - END statement does not exceed 27.

Error No.	Details	Remedy
P294	DO and END not paired The DO's and END's are not paired off properly.	<ul style="list-style-type: none"> Reconsider the program and correct it so that the DO's and END's are paired off properly.
P295	WHILE/GOTO in tape There is a WHILE or GOTO statement on the tape during tape operation.	<ul style="list-style-type: none"> During tape operation, a program which includes a WHILE or GOTO statement cannot be executed and so the memory operation mode is established instead.
P296	No address (macro) A required address has not been specified in the user macro.	<ul style="list-style-type: none"> Review the program.
P297	Address-A error The user macro does not use address A as a variable.	<ul style="list-style-type: none"> Review the program.
P298	G200-G202 cmdnd in tape User macro G200, G201, or G202 was specified during tape or MDI mode.	<ul style="list-style-type: none"> Review the program.
P300	Variable name illegal The variable names have not been commanded properly.	<ul style="list-style-type: none"> Reconsider the variable names in the program and correct them.
P301	Variable name duplicated The name of the variable has been duplicated.	<ul style="list-style-type: none"> Correct the program so that the name is not duplicated.
P310	Not use GMSTB macro code G, M, S, T, or B macro code was called during fixed cycle.	<ul style="list-style-type: none"> Review the program. Review the parameter.
P350	No spec: Scaling command The scaling command (G50, G51) was issued when the scaling specifications were not available.	<ul style="list-style-type: none"> Check the specifications.
P360	No spec: Program mirror A mirror image (G50.1 or G51.1) command has been issued though the programmable mirror image specifications are not provided.	<ul style="list-style-type: none"> Check the specifications.
P370	No spec: Facing t-post MR The facing turret mirror image specifications are not provided.	<ul style="list-style-type: none"> Check the specifications.
P371	Facing t-post MR illegal Mirror image for facing tool posts was commanded to an axis for which external mirror image or parameter mirror image is valid. Mirror image for facing tool posts validating mirror image for a rotary axis was commanded.	<ul style="list-style-type: none"> Check the program. Check the parameters.
P380	No spec: Corner R/C The corner R/C was issued when the corner R/C specifications were not available.	<ul style="list-style-type: none"> Check the specifications. Remove the corner chamfering/corner rounding command from the program.

Error No.	Details	Remedy
P381	No spec: Arc R/C Corner chamfering II /corner rounding II was specified in the arc interpolation block although corner chamfering/corner rounding II is unsupported.	<ul style="list-style-type: none"> Check the specifications.
P382	No corner movement The block next to corner chamfering/corner rounding is not a movement command.	<ul style="list-style-type: none"> Replace the block succeeding the corner chamfering/corner rounding command by G01 command.
P383	Corner movement short In the corner chamfering/corner rounding command, the movement distance was shorter than the value in the corner chamfering/corner rounding command.	<ul style="list-style-type: none"> Make the corner chamfering/corner rounding less than the movement distance since this distance is shorter than the corner chamfering/corner rounding.
P384	Corner next movement short When the corner chamfering/corner rounding command was input, the movement distance in the following block was shorter than the length of the corner chamfering/corner rounding.	<ul style="list-style-type: none"> Make the corner chamfering/corner rounding less than the movement distance since this distance in the following block is shorter than the corner chamfering/corner rounding.
P385	Corner during G00/G33 A block with corner chamfering/corner rounding was given during G00 or G33 modal.	<ul style="list-style-type: none"> Recheck the program.
P390	No spec: Geometric A geometric command was issued though there are no geometric specifications.	<ul style="list-style-type: none"> Check the specifications.
P391	No spec: Geometric arc There are no geometric IB specifications.	<ul style="list-style-type: none"> Check the specifications.
P392	Angle < 1 degree (GEOMT) The angular difference between the geometric line and line is 1° or less.	<ul style="list-style-type: none"> Correct the geometric angle.
P393	Inc value in 2nd block (GEOMT) The second geometric block was specified by an incremental value.	<ul style="list-style-type: none"> Specify this block by an absolute value.
P394	No linear move command (GEOMT) The second geometric block contains no linear command.	<ul style="list-style-type: none"> Specify the G01 command.
P395	Illegal address (GEOMT) The geometric format is invalid.	<ul style="list-style-type: none"> Recheck the program.
P396	Plane selected in GEOMT ctrl A plane switching command was executed during geometric command processing.	<ul style="list-style-type: none"> Execute the plane switching command before geometric command processing.
P397	Arc error (GEOMT) In geometric IB, the circular arc end point does not contact or cross the next block start point.	<ul style="list-style-type: none"> Recheck the geometric circular arc command and the preceding and following commands.
P398	No spec: Geometric1B Although the geometric IB specifications are not included, a geometric command is given.	<ul style="list-style-type: none"> Check the specifications.

Error No.	Details	Remedy
P411	Illegal modal G111 <ul style="list-style-type: none"> • G111 was issued during milling mode. • G111 was issued during nose R compensation mode. • G111 was issued during constant surface speed. • G111 was issued during mixed synchronization control. • G111 was issued during fixed cycle. • G111 was issued during polar coordinate interpolation. • G111 was issued during cylindrical interpolation mode. 	<ul style="list-style-type: none"> • Before commanding G111, cancel the following commands. <ul style="list-style-type: none"> • Milling mode • Nose R compensation • Constant surface speed • Mixed synchronization control • Fixed cycle • Polar coordinate interpolation • Cylindrical interpolation
P412	P412 No spec: Axis name switch Axis name switch (G111) was issued even though there is no axis name switch (G111) specification.	<ul style="list-style-type: none"> • Check the specifications.
P420	No spec: Para input by program Parameter input by program (G10) was commanded even though there is no specification of parameter input by program.	<ul style="list-style-type: none"> • Check the specifications.
P421	Parameter input error <ul style="list-style-type: none"> • The specified parameter No. or set data is illegal. • An illegal G command address was input in parameter input mode. • A parameter input command was input during fixed cycle modal or nose R compensation. • G10L50, G10L70, G11 were not commanded in independent blocks. 	<ul style="list-style-type: none"> • Check the program.
P430	R-pnt return incomplete <ul style="list-style-type: none"> • A command was issued to move an axis, which has not returned to the reference position, away from that reference position. • A command was issued to an axis removal axis. 	<ul style="list-style-type: none"> • Execute reference position return manually. • The command was issued to an axis for which axis removal is validated so invalidate axis removal.
P431	No spec: 2,3,4th R-point ret A command for second, third or fourth reference position return was issued though there are no such command specifications.	<ul style="list-style-type: none"> • Check the specifications.
P432	No spec: Start position return Start position return (G29) was commanded even though there is no start position return specification.	<ul style="list-style-type: none"> • Check the specifications.
P433	No spec: R-position check Reference position check (G27) was commanded even though there is no reference position check specification.	<ul style="list-style-type: none"> • Check the specifications.

Error No.	Details	Remedy
P434	Compare error One of the axes did not return to the reference position when the reference position check command (G27) was executed.	<ul style="list-style-type: none"> • Check the program.
P435	G27 and M commands in a block An M command was issued simultaneously in the G27 command block.	<ul style="list-style-type: none"> • An M code command cannot be issued in a G27 command block and so the G27 command and M code command must be placed in separate blocks.
P436	G29 and M commands in a block An M command was issued simultaneously in the G29 command block.	<ul style="list-style-type: none"> • An M code command cannot be issued in a G29 command block and so the G29 command and M code command must be placed in separate blocks.
P438	G52 invalid during G54.1 A local coordinate system command was issued during execution of the G54.1 command.	<ul style="list-style-type: none"> • Review the program.
P450	No spec: Chuck barrier The chuck barrier on command (G22) was specified although the chuck barrier was undefined in the specification.	<ul style="list-style-type: none"> • Check the specification.
P451	No spec: Stroke chk bef travel Stroke check before travel (G22/G23) was commanded even though there is no stroke check before travel specification.	<ul style="list-style-type: none"> • Check the specification.
P452	Limit before travel exists An illegal command such as the start or end point of the traveling axis is inside the prohibited area or the axis passes through the prohibited area, was detected when Stroke check before travel (G22) was ON.	<ul style="list-style-type: none"> • Review the coordinate values of the axis address commanded in the program.
P460	Tape I/O error An error has arisen in the tape reader or, alternatively, in the printer during macro printing.	<ul style="list-style-type: none"> • Check the power and cable of the connected devices. • Check the I/O device parameters.
P461	File I/O error A file of the machining program cannot be read. IC card has not been inserted.	<ul style="list-style-type: none"> • In memory mode, the programs stored in memory may have been destroyed. Output all of the programs and tool data once and format them. • Ensure that the external device (including an IC card, etc) that contains the file is mounted. • Check the parameters for HD operation or IC card operation.
P462	Computer link commu error A communication error occurred during the BTR operation.	<ul style="list-style-type: none"> • "L01 Computer link error" is displayed simultaneously, so remedy the problem according to the error No.
P480	No spec: Milling <ul style="list-style-type: none"> • Milling was commanded when the milling specifications were not provided. • Polar coordinate interpolation was commanded when the polar coordinate interpolation specifications were not provided. 	<ul style="list-style-type: none"> • Check the specification.

Error No.	Details	Remedy
P481	Illegal G code (mill) <ul style="list-style-type: none"> • An illegal G code was used during the milling mode. • An illegal G code was used during cylindrical interpolation or polar coordinate interpolation. • The G07.1 command was issued during the tool radius compensation. 	<ul style="list-style-type: none"> • Check the program.
P482	Illegal axis (mill) <ul style="list-style-type: none"> • A rotary axis was commanded during the milling mode. • Milling was executed even though an illegal value was set for the milling axis No. • Cylindrical interpolation or polar coordinate interpolation was commanded during mirror image. • Cylindrical interpolation or polar coordinate interpolation was commanded before the tool compensation was completed after the T command. • G07.1 was commanded when cylindrical interpolation was not possible (there is no rotary axis, or external mirror image is ON). • An axis other than a cylindrical coordinate system axis was commanded during cylindrical interpolation. 	<ul style="list-style-type: none"> • Check the machining program, parameters and PLC I/F signal.
P484	R-pnt ret incomplete (mill) <ul style="list-style-type: none"> • Movement was commanded to an axis that had not completed reference position return during the milling mode. • Movement was commanded to an axis that had not completed reference position return during cylindrical interpolation or polar coordinate interpolation. 	<ul style="list-style-type: none"> • Carry out manual reference position return.

Error No.	Details	Remedy
P485	Illegal modal (mill) <ul style="list-style-type: none"> • The milling mode was turned ON during nose R compensation or constant surface speed control. • A T command was issued during the milling mode. • The mode was switched from milling to cutting during tool compensation. • Cylindrical interpolation or polar coordinate interpolation was commanded during the constant surface speed control mode (G96). • The command unacceptable in the cylindrical interpolation was issued. • A T command was issued during the cylindrical interpolation or polar coordinate interpolation mode. • A movement command was issued when the plane was not selected just before or after the G07.1 command. • A plane selection command was issued during the polar coordinate interpolation mode. • Cylindrical interpolation or polar coordinate interpolation was commanded during tool radius compensation. • The G16 plane in which the radius value of a cylinder is 0 was specified. • A cylindrical interpolation or polar coordinate interpolation command was issued during coordinate rotation by program (G68). 	<ul style="list-style-type: none"> • Check the program. • Before issuing G12.1, issue G40 or G97. • Before issuing G12.1, issue a T command. • Before issuing G13.1, issue G40. • Specify the radius value of a cylinder other than 0, or specify the X axis's current value other than 0 before issuing G12.1/G16.
P486	Milling error <ul style="list-style-type: none"> • The milling command was issued during the mirror image (when parameter or external input is turned ON). • Polar coordinate interpolation, cylindrical interpolation or milling interpolation was commanded during mirror image for facing tool posts. • The start command of the cylindrical interpolation or polar coordinate interpolation was issued during the normal line control. 	<ul style="list-style-type: none"> • Check the program.
P511	Synchronization M code error <ul style="list-style-type: none"> • Two or more synchronization M codes were commanded in the same block. • The synchronization M code and "!" code were commanded in the same block. • Synchronization with the M code was commanded in 3rd part system or more. (Synchronization with the M code is valid only in 1st part system or 2nd part system.) 	<ul style="list-style-type: none"> • Check the program.
P550	No spec: G06.2(NURBS) There is no NURBS interpolation option.	<ul style="list-style-type: none"> • Check the specifications.

Error No.	Details	Remedy
P551	G06.2 knot error The knot (k) command value is smaller than the value for the previous block.	<ul style="list-style-type: none"> Reconsider the program. Specify the knot by monotone increment.
P552	Start point of 1st G06.2 err The block end point immediately before the G06.2 command and the G06.2 first block command value do not match.	<ul style="list-style-type: none"> Match the G06.2 first block coordinate command value with the previous block end point.
P554	Invld manual interrupt in G6.2 Manual interruption using a block was performed while in G06.2 mode.	<ul style="list-style-type: none"> Perform for blocks other than G06.2 mode when manually interrupting.
P555	Invalid restart during G06.2 Restart was attempted from the block in G06.2 mode.	<ul style="list-style-type: none"> Restart from the block other than in G06.2 mode.
P600	No spec: Auto TLM An automatic tool length measurement command (G37) was execute though there are no such command specifications.	<ul style="list-style-type: none"> Check the specifications.
P601	No spec: Skip A skip command (G31) was issued though there are no such command specifications.	<ul style="list-style-type: none"> Check the specifications.
P602	No spec: Multi skip A multiple skip command (G31.1, G31.2 or G31.3) was issued though there are no such command specifications.	<ul style="list-style-type: none"> Check the specifications.
P603	Skip speed 0 The skip speed is 0.	<ul style="list-style-type: none"> Specify the skip speed.
P604	TLM illegal axis No axis or more than one axis was specified in the automatic tool length measurement block.	<ul style="list-style-type: none"> Specify only one axis.
P605	T & TLM command in a block The T code is in the same block as the automatic tool length measurement block.	<ul style="list-style-type: none"> Specify this T code before the block.
P606	T cmdnd not found before TLM The T code was not yet specified in automatic tool length measurement.	<ul style="list-style-type: none"> Specify this T code before the block.
P607	TLM illegal signal Before the area specified by the D command or decelerating area parameter d, the measurement position arrival signal went ON. The signal remains OFF to the end.	<ul style="list-style-type: none"> Check the program.
P608	Skip during radius compen A skip command was specified during radius compensation processing.	<ul style="list-style-type: none"> Specify a radius compensation cancel (G40) command or remove the skip command.
P610	Illegal parameter <ul style="list-style-type: none"> The parameter setting is not correct. G114.1 was commanded when the spindle synchronization with PLC I/F command was selected. G113 was commanded when the spindle-spindle polygon machining option was OFF and the spindle synchronization with PLC I/F command was selected. 	<ul style="list-style-type: none"> Check whether "#1549 lv0vR1" to "#1553 lv0vR5" are set in descending order (in order of large values). Check whether "#1554 lv0rd2" to "#1557 lv0rd5" are set in descending order. Check and correct "#1514 expLinax" and "#1515 expRotax". Check the program. Check the parameter.

Error No.	Details	Remedy
P611	No spec: Exponential function Specification for exponential interpolation is not available.	<ul style="list-style-type: none"> • Check the specification.
P612	Exponential function error A movement command for exponential interpolation was issued during mirror image for facing tool posts.	<ul style="list-style-type: none"> • Check the program.
P700	Illegal command value Spindle synchronization was commanded to a spindle that is not connected serially.	<ul style="list-style-type: none"> • Check the program. • Check the parameter.
P900	No spec: Normal line control A normal line control command (G40.1, G41.1, G42.1) was issued when the normal line control specifications were not provided.	<ul style="list-style-type: none"> • Check the specifications.
P901	Normal line control axis G92 A coordinate system preset command (G92) was issued to a normal line control axis during normal line control.	<ul style="list-style-type: none"> • Check the program.
P902	Normal line control axis error <ul style="list-style-type: none"> • The normal line control axis was set to a linear axis. • The normal line control axis was set to the linear type rotary axis II axis. • The normal line control axis has not been set. • The normal line control axis was the same as the plane selection axis. 	<ul style="list-style-type: none"> • Correct the normal line control axis.
P903	Plane chg in Normal line ctrl The plane selection command (G17, G18, G19) was issued during normal line control.	<ul style="list-style-type: none"> • Delete the plane selection command (G17, G18, G19) from the program for normal line control.
P920	No spec: 3D coord conv There is no specification for 3-dimensional coordinate conversion.	<ul style="list-style-type: none"> • Check the specifications.
P921	Illegal G code at 3D coord A G code command that cannot be performed was made during 3-dimensional coordinate conversion modal.	<ul style="list-style-type: none"> • Refer to "Mitsubishi CNC 700/70 Series Programming Instruction Manual (Machining Center Series)" for further details of usable G commands. • When the basic specification parameter "#1229 set01/bit3" is ON, turn the parameter OFF or specify the constant surface speed control cancel (G97).
P922	Illegal mode at 3D coord A 3-dimensional coordinate conversion command was issued during a modal for which 3-dimensional coordinate conversion cannot be performed.	<ul style="list-style-type: none"> • Refer to "Mitsubishi CNC 700/70 Series Programming Instruction Manual (Machining Center Series)" for further details of usable G commands.
P923	Illegal addr in 3D coord blk A G code for which G68 to combination could not be performed was specified for the same block.	<ul style="list-style-type: none"> • Refer to "Mitsubishi CNC 700/70 Series Programming Instruction Manual (Machining Center Series)" for further details of usable G commands.

Error No.	Details	Remedy
P930	No spec: Tool axis compen A tool length compensation along the tool axis command was issued even though there is no tool length compensation along the tool axis specification.	<ul style="list-style-type: none"> • Check the specifications.
P931	Executing tool axis compen A G code that cannot be commanded exists during tool length compensation along the tool axis.	<ul style="list-style-type: none"> • Reconsider the program.
P932	Rot axis parameter error There is a mistake in the linear axis name and rotary axis name in the rotary axis configuration parameters.	<ul style="list-style-type: none"> • Set the correct value and reboot.
P940	No spec: Tool tip control There is no tool tip center control specification.	<ul style="list-style-type: none"> • Check the specifications.
P941	Invalid T tip control command A tool tip center control command was issued during a modal for which a tool tip center control command cannot be issued.	<ul style="list-style-type: none"> • Reconsider the program.
P942	Invalid cmd during T tip ctrl A G code that cannot be commanded was issued during tool tip center control.	<ul style="list-style-type: none"> • Reconsider the program.
P943	Tool posture command illegal In the case of tool tip center control type 1, if the signs at the tool-side rotary axis or table base-side rotary axis start and finish points differ, a tool base-side rotary axis or table workpiece-side rotary axis rotation exists for the same block, and does not pass a singular point. In the case of tool tip center control type 2, the posture vector command is incorrect.	<ul style="list-style-type: none"> • Reconsider the program.
P990	PREPRO error Combining commands that required pre-reading (nose R offset, corner chamfering/corner rounding, geometric I, geometric IB, and compound type fixed cycle for turning machining) resulted in eight or more pre-read blocks.	<ul style="list-style-type: none"> • Reduce the number of commands that require pre-reading or delete such commands.

Appendix 7. G Code Guidance Display List

<M system>

O: Modal, x: Unmodal

G code	Group	Function	Modal
00	01	Positioning	O
01	01	Linear interpolation	O
02	01	Circular interpolation CW (clockwise) / Spiral/Conical interpolation CW (type2)	O
03	01	Circular interpolation CCW (counterclockwise) / Spiral/Conical interpolation CCW (type2)	O
02.1	01	Spiral/Conical interpolation CW (type1)	O
03.1	01	Spiral/Conical interpolation CCW (type1)	O
02.3	01	Exponential function interpolation positive rotation	O
03.3	01	Exponential function interpolation negative rotation	O
02.4	01	3-dimensional circular interpolation	O
03.4	01	3-dimensional circular interpolation	O
04	00	Dwell	x
05	00	High-speed high-accuracy control II/High-speed machining mode	O
05.1	00	High-speed high-accuracy control I/Spline	O
06.2	01	NURBS interpolation	O
07	00	Hypothetical axis interpolation	O
07.1 107	21	Cylindrical interpolation	O
08	00	High-accuracy control 1	O
09	00	Exact stop check	x
10	00	Program data input (parameter /compensation data/parameter coordinate rotation data)	-
11	00	Program data input cancel	-
12	00	Circular cut CW (clockwise)	x
13	00	Circular cut CCW (counterclockwise)	x
12.1 112	21	Polar coordinate interpolation ON	O
13.1 113	21	Polar coordinate interpolation cancel	-
15	18	Polar coordinate command OFF	-
16	18	Polar coordinate command ON	O
17	02	Plane selection X-Y	O
18	02	Plane selection Z-X	O
19	02	Plane selection Y-Z	O
20	06	Inch command	O
21	06	Metric command	O
22	04	Stroke check before travel ON	-
23	04	Stroke check before travel cancel	-
27	00	Reference position check	x
28	00	Reference position return	x
29	00	Start position return	x
30	00	2nd to 4th reference position return	x
30.1	00	Tool change position return 1	x
30.2	00	Tool change position return 2	x
30.3	00	Tool change position return 3	x
30.4	00	Tool change position return 4	x
30.5	00	Tool change position return 5	x
30.6	00	Tool change position return 6	x

Appendix 7. G Code Guidance Display List

O: Modal, x: Unmodal

G code	Group	Function	Modal
31	00	Skip/Multi-step skip function 2	x
31.1	00	Multi-step skip function 1-1	x
31.2	00	Multi-step skip function 1-2	x
31.3	00	Multi-step skip function 1-3	x
33	01	Thread cutting	O
34	00	Special fixed cycle (bolt hole circle)	x
35	00	Special fixed cycle (line at angle)	x
36	00	Special fixed cycle (arc)	x
37	00	Automatic tool length measurement	x
37.1	00	Special fixed cycle (grid)	x
38	00	Tool radius compensation vector designation	x
39	00	Tool radius compensation corner arc	x
40	07	Tool radius compensation cancel / 3-dimentional tool radius compensation cancel	-
41	07	Tool radius compensation left / 3-dimentional tool radius compensation left	O
42	07	Tool radius compensation right / 3-dimentional tool radius compensation right	O
40.1	15	Normal line control cancel	-
41.1	15	Normal line control left ON	O
42.1	15	Normal line control right ON	O
43	08	Tool length compensation (+)	O
44	08	Tool length compensation (-)	O
43.1	08	Tool length compensation along the tool axis	O
43.4	08	Tool center point control type 1	O
43.5	08	Tool center point control type 2	O
45	00	Tool position offset (extension)	x
46	00	Tool position offset (reduction)	x
47	00	Tool position offset (doubled)	x
48	00	Tool position offset (halved)	x
49	08	Tool length compensation cancel/Tool center point control cancel	-
50	11	Scaling cancel	-
51	11	Scaling ON	O
50.1	19	G command mirror image cancel	-
51.1	19	G command mirror image ON	O
52	00	Local coordinate system setting	O
53	00	Basic machine coordinate system selection	x
54	12	Workpiece coordinate system 1 selection	O
55	12	Workpiece coordinate system 2 selection	O
56	12	Workpiece coordinate system 3 selection	O
57	12	Workpiece coordinate system 4 selection	O
58	12	Workpiece coordinate system 5 selection	O
59	12	Workpiece coordinate system 6 selection	O
54.1	12	Workpiece coordinate system selection 48 / 96 sets extended	O
60	00	Unidirectional positioning	x
61	13	Exact stop check mode	O
61.1	13	High-accuracy control 1 ON	O
61.2	13	High-accuracy spline interpolation	O
62	13	Automatic corner override	O
63	13	Tapping mode	O
63.1	13	Synchronous tapping mode (normal tapping)	O
63.2	13	Synchronous tapping mode (reverse tapping)	O

Appendix 7. G Code Guidance Display List

O: Modal, x: Unmodal

G code	Group	Function	Modal
64	13	Cutting mode	O
65	00	User macro call	x
66	14	User macro modal call A	O
66.1	14	User macro modal call B	O
67	14	User macro modal call cancel	-
68	16	Programmable coordinate rotation mode ON/3-dimensional coordinate conversion mode ON	O
69	16	Programmable coordinate rotation mode OFF/3-dimensional coordinate conversion mode OFF	-
73	09	Fixed cycle (step)	O
74	09	Fixed cycle (reverse tap)	O
75	09	Fixed cycle (circle cutting cycle)	O
76	09	Fixed cycle (fine boring)	O
80	09	Fixed cycle cancel	-
81	09	Fixed cycle (drill/spot drill)	O
82	09	Fixed cycle (drill/counter boring)	O
83	09	Fixed cycle (deep drilling)	O
84	09	Fixed cycle (tapping)	O
85	09	Fixed cycle (boring)	O
86	09	Fixed cycle (boring)	O
87	09	Fixed cycle (back boring)	O
88	09	Fixed cycle (boring)	O
89	09	Fixed cycle (boring)	O
90	03	Absolute value command	O
91	03	Incremental command value	O
92	00	Coordinate system setting	O
92.1	00	Workpiece coordinate system pre-setting	x
93	05	Inverse time feed	O
94	05	Per-minute feed (Asynchronous feed)	O
95	05	Per-revolution feed (Synchronous feed)	O
96	17	Constant surface speed control ON	O
97	17	Constant surface speed control OFF	-
98	10	Fixed cycle Initial level return	O
99	10	Fixed cycle R point level return	O

Appendix 7. G Code Guidance Display List

<L system>

O: Modal, x: Unmodal

G code list						Group	Function	Modal
2	3	4	5	6	7			
G00	G00	G00	G00	G00	G00	01	Positioning	O
G01	G01	G01	G01	G01	G01	01	Linear interpolation	O
G02	G02	G02	G02	G02	G02	01	Circular interpolation CW / Helical interpolation CW	O
G03	G03	G03	G03	G03	G03	01	Circular interpolation CCW / Helical interpolation CCW	O
G02.3	G02.3	G02.3	G02.3	G02.3	G02.3	01	Exponential interpolation CW	O
G03.3	G03.3	G03.3	G03.3	G03.3	G03.3	01	Exponential interpolation CCW	O
G04	G04	G04	G04	G04	G04	00	Dwell	x
				G07.1 G107	G07.1 G107	19	Cylindrical interpolation	O
G09	G09	G09	G09	G09	G09	00	Exact stop check	x
G10	G10	G10	G10	G10	G10	00	Parameter/Compensation data input by program/ Tool life management data registration	-
G11	G11	G11	G11	G11	G11	00	Program parameter input / Tool life management data registration mode cancel	-
				G12.1 G112	G12.1 G112	19	Polar coordinate interpolation ON	O
				G13.1 G113	G13.1 G113	19	Polar coordinate interpolation cancel	-
G12.1	G12.1	G12.1	G12.1			19	Milling interpolation ON	O
G13.1	G13.1	G13.1	G13.1			19	Milling interpolation cancel	-
G14	G14	G14	G14			18	Balance cut OFF	-
G15	G15	G15	G15			18	Balance cut ON	O
G16	G16	G16	G16			02	Milling interpolation plane selection Y-Z cylindrical plane	O
G17	G17	G17	G17	G17	G17	02	Plane selection X-Y	O
G18	G18	G18	G18	G18	G18	02	Plane selection Z-X	O
G19	G19	G19	G19	G19	G19	02	Plane selection Y-Z	O
G20	G20	G20	G20	G20	G20	06	Inch command	O
G21	G21	G21	G21	G21	G21	06	Metric command	O
G22	G22	G22	G22			04	Barrier check ON	O
G23	G23	G23	G23			04	Barrier check OFF	-
				G22	G22	00	Soft limit ON	x
				G23	G23	00	Soft limit OFF	-
G27	G27	G27	G27	G27	G27	00	Reference position return check	x
G28	G28	G28	G28	G28	G28	00	Automatic reference position return	x
G29	G29	G29	G29	G29	G29	00	Return from reference position	x
G30	G30	G30	G30	G30	G30	00	2nd, 3rd and 4th reference position return	x
G30.1	G30.1	G30.1	G30.1	G30.1	G30.1	00	Tool change position return 1	x
G30.2	G30.2	G30.2	G30.2			00	Tool change position return 2	x
G30.3	G30.3	G30.3	G30.3			00	Tool change position return 3	x
G30.4	G30.4	G30.4	G30.4			00	Tool change position return 4	x
G30.5	G30.5	G30.5	G30.5			00	Tool change position return 5	x
G31	G31	G31	G31	G31	G31	00	Skip function/Multiple-step skip function 2	x

Appendix 7. G Code Guidance Display List

O: Modal, x: Unmodal

G code list						Group	Function	Modal
2	3	4	5	6	7			
G31.1	G31.1	G31.1	G31.1	G31.1	G31.1	00	Multiple-step skip function 1-1	x
G31.2	G31.2	G31.2	G31.2	G31.2	G31.2	00	Multiple-step skip function 1-2	x
G31.3	G31.3	G31.3	G31.3	G31.3	G31.3	00	Multiple-step skip function 1-3	x
G32	G33	G32	G33	G32	G33	01	Thread cutting	O
G34	G34	G34	G34	G34	G34	01	Variable lead thread cutting	O
G35	G35	G35	G35	G35	G35	01	Circular thread cutting CW	O
G36	G36	G36	G36	G36	G36	01	Circular thread cutting CCW	O
G37	G37	G36/G37	G36/G37	G36/G37 G37.1 G37.2	G36/G37 G37.1 G37.2	00	Automatic tool length measurement	x
G40	G40	G40	G40	G40	G40	07	Tool nose R compensation cancel	-
G41	G41	G41	G41	G41	G41	07	Tool nose R compensation left	O
G42	G42	G42	G42	G42	G42	07	Tool nose R compensation right	O
G46	G46	G46	G46	G46	G46	07	Tool nose R compensation (direction automatically selected) ON	O
G43.1	G43.1	G43.1	G43.1	G43.1	G43.1	20	1st spindle control mode	O
G44.1	G44.1	G44.1	G44.1	G44.1	G44.1	20	Selected spindle control mode	O
G47.1	G47.1	G47.1	G47.1	G47.1	G47.1	20	All spindles simultaneous control mode	O
G50	G92	G50	G92	G50	G92	00	Coordinate system setting/Spindle clamp speed setting	O
G50.2	G50.2	G50.2	G50.2			11	Scaling cancel	-
G51.2	G51.2	G51.2	G51.2			11	Scaling ON	O
				G50.2 G250	G50.2 G250	00	Polygon machining mode cancel (spindle-tool axis synchronization)	-
				G51.2 G251	G51.2 G251	00	Polygon machining mode ON (spindle-tool axis synchronization)	x
G52	G52	G52	G52	G52	G52	00	Local coordinate system setting	O
G53	G53	G53	G53	G53	G53	00	Basic machine coordinate system selection	x
G54	G54	G54	G54	G54	G54	12	Workpiece coordinate system selection 1	O
G55	G55	G55	G55	G55	G55	12	Workpiece coordinate system selection 2	O
G56	G56	G56	G56	G56	G56	12	Workpiece coordinate system selection 3	O
G57	G57	G57	G57	G57	G57	12	Workpiece coordinate system selection 4	O
G58	G58	G58	G58	G58	G58	12	Workpiece coordinate system selection 5	O
G59	G59	G59	G59	G59	G59	12	Workpiece coordinate system selection 6	O
G54.1	G54.1	G54.1	G54.1	G54.1	G54.1	12	Workpiece coordinate system 48 sets expanded	O
G61	G61	G61	G61	G61	G61	13	Exact stop check mode	O
G62	G62	G62	G62	G62	G62	13	Automatic corner override	O
G63	G63	G63	G63	G63	G63	13/19	Tapping mode	O
G64	G64	G64	G64	G64	G64	13/19	Cutting mode	O
G65	G65	G65	G65	G65	G65	00	User macro call	x
G66	G66	G66	G66	G66	G66	14	User macro modal call A	O
G66.1	G66.1	G66.1	G66.1	G66.1	G66.1	14	User macro modal call B	O
G67	G67	G67	G67	G67	G67	14	User macro modal call cancel	-

Appendix 7. G Code Guidance Display List

O: Modal, x: Unmodal

G code list						Group	Function	Modal
2	3	4	5	6	7			
G68	G68	G68	G68			15	Mirror image for facing tool posts ON	O
G69	G69	G69	G69			15	Mirror image for facing tool posts OFF	-
				G68	G68	15	Mirror image for facing tool posts ON or balance cut mode ON	O
				G69	G69	15	Mirror image for facing tool posts OFF or balance cut mode cancel	-
G70	G70	G70	G70	G70	G70	09	Finishing cycle	O
G71	G71	G71	G71	G71	G71	09	Longitudinal rough cutting cycle	O
G72	G72	G72	G72	G72	G72	09	Face rough cutting cycle	O
G73	G73	G73	G73	G73	G73	09	Formed material rough cutting cycle	O
G74	G74	G74	G74	G74	G74	09	Face cut-off cycle	O
G75	G75	G75	G75	G75	G75	09	Longitudinal cut-off cycle	O
G76	G76	G76	G76	G76	G76	09	Compound thread cutting cycle	O
G76.1	G76.1	G76.1	G76.1	G76.1	G76.1	09	2-part system synchronous thread-cutting cycle (1)	O
G76.2	G76.2	G76.2	G76.2	G76.2	G76.2	09	2-part system synchronous thread-cutting cycle (2)	O
G90	G77	G90	G77	G90	G77	09	Longitudinal cutting fixed cycle	O
G92	G78	G92	G78	G92	G78	09	Thread cutting fixed cycle	O
G94	G79	G94	G79	G94	G79	09	Face cutting fixed cycle	O
G80	G80	G80	G80	G80	G80	09	Fixed cycle for drilling cancel	-
G81	G81	G81	G81	G81	G81	09	Fixed cycle (drill/spot drilling)	O
G82	G82	G82	G82	G82	G82	09	Fixed cycle (drill/counter boring)	O
G79	G83.2	G79	G83.2	G79	G83.2	09	Deep hole drilling cycle 2	O
G83	G83	G83	G83	G83	G83	09	Deep hole drilling cycle (Z axis)/ Small-diameter deep-hole drilling cycle	O
G83.1	G83.1	G83.1	G83.1	G83.1	G83.1	09	Stepping cycle	O
G84	G84	G84	G84	G84	G84	09	Tap cycle (Z axis)	O
G85	G85	G85	G85	G85	G85	09	Boring cycle (Z axis)	O
G87	G87	G87	G87	G87	G87	09	Deep hole drilling cycle (X axis)	O
G88	G88	G88	G88	G88	G88	09	Tap cycle (X axis)	O
G89	G89	G89	G89	G89	G89	09	Boring cycle (X axis)	O
G84.1	G84.1	G84.1	G84.1	G84.1	G84.1	09	Reverse tap cycle (Z axis)	O
G84.2	G84.2	G84.2	G84.2	G84.2	G84.2	09	Synchronous tapping cycle	O
G88.1	G88.1	G88.1	G88.1	G88.1	G88.1	09	Reverse tap cycle (X axis)	O
G50.3	G92.1	G50.3	G92.1	G50.3	G92.1	00	Workpiece coordinate preset	x
G96	G96	G96	G96	G96	G96	17	Constant surface speed control ON	O
G97	G97	G97	G97	G97	G97	17	Constant surface speed control OFF	-
G98	G94	G98	G94	G98	G94	05	Feed per minute (Asynchronous feed)	O
G99	G95	G99	G95	G99	G95	05	Feed per revolution (Synchronous feed)	O
-	G90	-	G90	-	G90	03	Absolute value command	O
-	G91	-	G91	-	G91	03	Incremental value command	O
-	G98	-	G98	-	G98	10	Fixed cycle initial return	O
-	G99	-	G99	-	G99	10	Fixed cycle R point return	O

Appendix 7. G Code Guidance Display List

O: Modal, x: Unmodal

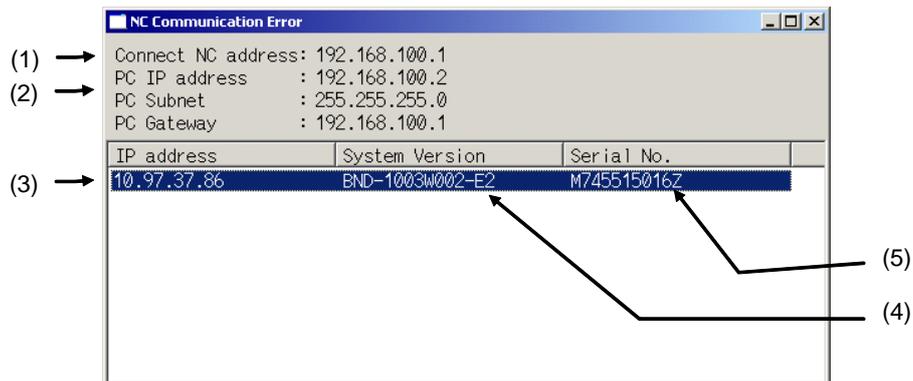
G code list						Group	Function	Modal
2	3	4	5	6	7			
G113	G113	G113	G113			00	Spindle synchronization polygon machining cancel (spindle-spindle synchronization) mode cancel	-
G114.1	G114.1	G114.1	G114.1			00	Spindle synchronization	x
G114.2	G114.2	G114.2	G114.2			00	Polygon machining (spindle-spindle synchronization) mode ON	x
G114.3	G114.3	G114.3	G114.3			00	Tool spindle synchronization II (Hobbing)	x
G115	G115	G115	G115	G115	G115	00	Start point designation synchronization Type 1	x
G116	G116	G116	G116	G116	G116	00	Start point designation synchronization Type 2	x
G117	G117	G117	G117	G117	G117	00	Miscellaneous function output during axis movement	x

Appendix 8. IP Address Resetting Procedure at Disabled Network Communication [700 Series Only]

8.1 Connectable Control Unit IP Address List Screen

After NC starts, when the communication between the control unit and the display unit cannot be established even after the time-out time passes, the connectable control unit IP address list screen appears. The contents of IP address list screen is displayed in English.

If no IP address is displayed, check if there are any loose or disconnected cable, hardware breakdown, etc.



Display items

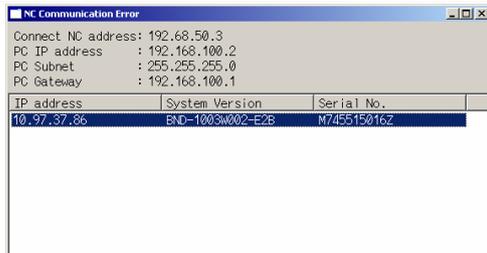
Display item	Details
(1) Connect NC address	This displays IP address of the connection destination control unit.
(2) PC IP address PC Subnet PC Gateway	This displays IP address, subnet mask, gateway settings.
(3) IP address	This displays IP address of the connectable control unit.
(4) System Version	This displays system version of the connectable control unit.
(5) Serial No.	This displays serial No. of the connectable control unit.

(Note 1) For (3), (4) and (5), up to ten lines are displayed at a time. When the display item exceeds ten lines, the following lines can be displayed with the **Page down** key.

8.2 Resetting Procedure

Operation method

- (1) Select IP address of the connection destination control unit by , , ,  keys from IP address list, and press the  key.



The dialog which notifies the change of the network setting is displayed as follows.



To establish the communication between the selected control unit and the display unit, the Ethernet parameter is temporarily changed.

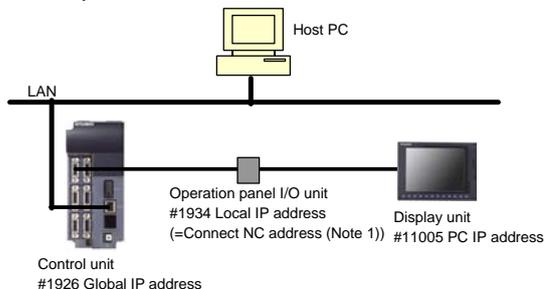
- (2) Press the  key.

After a while, the NC screen is displayed.
(If the NC screen is not displayed, remove the control unit from external network, and carry out the procedure (1) again.)

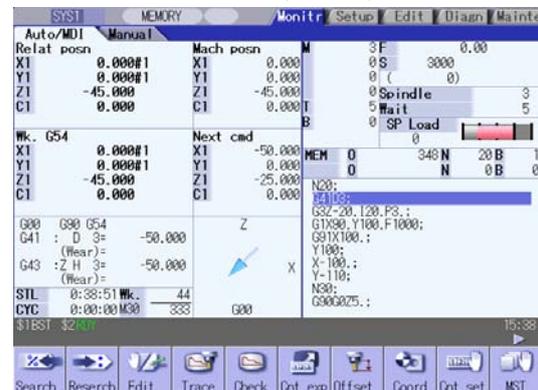
(Note) In the step, the communication between the control unit and the display unit is temporarily established. Therefore, unless the parameter is reset (in the procedure (3)), the system will be returned to the state of procedure (1) at the next NC startup.

- (3) Set the Ethernet parameter by referring to the following example of the network connection, and restart NC.

<Example of the network connection>



The NC screen is normally displayed.



(Note 1) "Connect NC address" is the same as the setting of "[HOSTS] TCP1" in setting file "melcfg.ini". The setting file "melcfg.ini" is in the following directory.

<FCU7-DA3xx-xx, FCU7-DA4xx-xx>

C:\¥WINDOWS¥melcfg.ini

<FCU7-DA2xx-xx>

¥ncsys¥melcfg.ini

(Note 2) When the control unit is added in user's network environment, match and change the parameters "#1926(PR) Global IP address", "#1927(PR) Global Subnet mask" and "#1928(PR) Global Gateway" to the user environment. Normally, the parameters "#1934 Local IP address", "#1935 Local Subnet mask" and "Connect NC address" need not be changed from the default setting value.

8.3 Message

The following messages display when IP address is reset.

Message	Details
Searching	<ul style="list-style-type: none"> The system is searching the connectable NC control unit to establish the communication between the control unit and the display unit. Please wait until the search is completed.
Socket error	<ul style="list-style-type: none"> The system could not find the connectable control unit because of the network interference. Turn OFF the NC power supply, and review the wiring for the network connection.
Setting error - Connect NC address	<ul style="list-style-type: none"> The automatic connection with the control unit failed because the setting file to set and save the unit's IP address was not found/is read-only/has illegal format. Review "[HOSTS] TCP1" setting in the setting file "melcfg.ini".
Searching IP address	<ul style="list-style-type: none"> The system is automatically searching the appropriate IP address because IP address of the display unit is inapposite. Please wait until the search is completed.
Review the setting of the Ethernet parameter after the screen starts.	<ul style="list-style-type: none"> The IP address setting of connected control unit and display unit was completed. Please review the setting of the Ethernet parameter after the screen starts.
Searching PC IP address failed.	<ul style="list-style-type: none"> No empty IP address was found by the automatic search for the IP address of the display unit. Please remove the control unit from the network, and turn ON the NC power supply again.

Appendix 9. User Parameter List

9.1 Process Parameters

 [#1026]	 base_I	 Base axis I
<p>Set the names of the basic axes that compose the plane. Set the axis name set in "#1013 axname". If all three items ("base_I", "base_J" and "base_K") do not need to be set, such as for 2-axis specifications, input "0", and the parameter will be blank. Normally, when X, Y and Z are specified respectively for base_I,_J,_K, the following relation will be established: G17: X-Y G18: Z-X G19: Y-Z Specify the desired axis name to set an axis address other than above.</p>		
<p>---Setting range--- Axis names such as X, Y or Z</p>		
 [#1027]	 base_J	 Base axis J
<p>Set the names of the basic axes that compose the plane. Set the axis name set in "#1013 axname". If all three items ("base_I", "base_J" and "base_K") do not need to be set, such as for 2-axis specifications, input "0", and the parameter will be blank. Normally, when X, Y and Z are specified respectively for base_I,_J,_K, the following relation will be established: G17: X-Y G18: Z-X G19: Y-Z Specify the desired axis name to set an axis address other than above.</p>		
<p>---Setting range--- Axis names such as X, Y or Z</p>		
 [#1028]	 base_K	 Base axis K
<p>Set the names of the basic axes that compose the plane. Set the axis name set in "#1013 axname". If all three items ("base_I", "base_J" and "base_K") do not need to be set, such as for 2-axis specifications, input "0", and the parameter will be blank. Normally, when X, Y and Z are specified respectively for base_I,_J,_K, the following relation will be established: G17: X-Y G18: Z-X G19: Y-Z Specify the desired axis name to set an axis address other than above.</p>		
<p>---Setting range--- Axis names such as X, Y or Z</p>		
 [#1029]	 aux_I	 Flat axis I
<p>Set the axis name when there is an axis parallel to "#1026 base_I".</p>		
<p>---Setting range--- Axis names such as X, Y or Z</p>		
 [#1030]	 aux_J	 Flat axis J
<p>Set the axis name when there is an axis parallel to "#1027 base_J".</p>		
<p>---Setting range--- Axis names such as X, Y or Z</p>		
 [#1031]	 aux_K	 Flat axis K
<p>Set the axis name when there is an axis parallel to "#1028 base_K".</p>		
<p>---Setting range--- Axis names such as X, Y or Z</p>		

- 【#1084】 RadErr Arc error**
Set the tolerable error range when the end point deviates from the center coordinate in the circular command.
---Setting range---
0 to 1.000 (mm)
- 【#1171】 taprov Tap return ovr**
Set the tap return override value for the synchronous tapping.
When "0" is set, it will be regarded as 100%.
---Setting range---
1 to 100 (%)
- 【#1185】 spd_F1 Feedrate F1**
Set the feedrate for the F command in the F 1-digit command ("#1079 F1digit" is set to "1").
Feedrate when F1 is issued (mm/min)
---Setting range---
1 to 60000 (mm/min)
- 【#1186】 spd_F2 Feedrate F2**
Set the feedrate for the F command in the F 1-digit command ("#1079 F1digit" is set to "1").
Feedrate when F2 is issued (mm/min)
---Setting range---
1 to 60000 (mm/min)
- 【#1187】 spd_F3 Feedrate F3**
Set the feedrate for the F command in the F 1-digit command ("#1079 F1digit" is set to "1").
Feedrate when F3 is issued (mm/min)
---Setting range---
1 to 60000 (mm/min)
- 【#1188】 spd_F4 Feedrate F4**
Set the feedrate for the F command in the F 1-digit command ("#1079 F1digit" is set to "1").
Feedrate when F4 is issued (mm/min)
---Setting range---
1 to 60000 (mm/min)
- 【#1189】 spd_F5 Feedrate F5**
Set the feedrate for the F command in the F 1-digit command ("#1079 F1digit" is set to "1").
Feedrate when F5 is issued (mm/min)
---Setting range---
1 to 60000 (mm/min)
- 【#1506】 F1_FM F1 upper limit**
Set the maximum value up to which the F 1-digit feedrate can be changed.
---Setting range---
0 to 60000 (mm/min)
- 【#1507】 F1_K F1 change constant**
Set the constant that determines the speed change rate per manual handle graduation in F 1-digit feedrate change mode.
---Setting range---
0 to 32767

<WRK COUNT> (No. of workpieces machined)

【#8001】 WRK COUNT M

Set the M code for counting the number of the workpiece repeated machining.
The number of the M-codes set by this parameter is counted.
The No. will not be counted when set to "0".

---Setting range---

0 to 99

【#8002】 WRK COUNT

Set the initial value of the number of workpiece machining. The number of current workpiece machining is displayed.

---Setting range---

0 to 999999

【#8003】 WRK COUNT LIMIT

Set the maximum number of workpiece machining.
A signal will be output to PLC when the number of machining times is counted to this limit.

---Setting range---

0 to 999999

<AUTO TLM> (Automatic tool length measurement)

【#8004】 SPEED

Set the feedrate during automatic tool length measurement.

---Setting range---

1 to 1000000 (mm/min)

【#8005】 ZONE r

Set the distance between the measurement point and deceleration start point.

---Setting range---

0 to 99999.999 (mm)

【#8006】 ZONE d

Set the tolerable range of the measurement point.
An alarm will occur when the sensor signal turns ON before the range, set by this parameter, has not been reached from the measurement point, or when the signal does not turn ON after the range is passed.

---Setting range---

0 to 99999.999 (mm)

<AUTO CORNER OVR.> (Automatic corner override)

【#8007】 OVERRIDE

Set the override value for automatic corner override.

---Setting range---

0 to 100 (%)

【#8008】 MAX ANGLE

Set the maximum corner opening angle where deceleration should start automatically.
When the angle is larger than this value, deceleration will not start.

---Setting range---

0 to 180 (°)

【#8009】 DSC. ZONE

Set the position where deceleration starts at the corner.
Designate at which length point before the corner deceleration should start.

---Setting range---

0 to 99999.999 (mm)

<T-TIP OFFSET> (Wear data input)**【#8010】 ABS. MAX for L system only**

Set the maximum value when inputting the tool wear compensation amount.
 A value exceeding this setting value cannot be set.
 Absolute value of the input value is set.
 (If a negative value is input, it is treated and set as a positive value.)
 If "0" is input, this parameter will be disabled.

---Setting range---

0 to 99.999 (mm)
 (Input setting increment applies)

【#8011】 INC. MAX for L system only

Set the maximum value for when inputting the tool wear compensation amount in the incremental mode.
 A value exceeding this setting value cannot be set.
 Absolute value of the input value is set.
 (If a negative value is input, it is treated and set as a positive value.)
 If "0" is input, this parameter will be disabled.

---Setting range---

0 to 99.999 (mm)
 (Input setting increment applies)

<C-MODAL> (C axis normal line)**【#8041】 C-rot.R**

Set the length from the center of the normal line control axis to the tool tip. This is used to calculate the turning speed at the block joint.
 This is enabled during the normal line control type II.

---Setting range---

0.000 to 99999.999 (mm)

【#8042】 C-ins.R

Set the radius of the arc to be automatically inserted into the corner during normal line control.
 This is enabled during the normal line control type I.

---Setting range---

0.000 to 99999.999 (mm)

【#8043】 Tool HDL FD OFS

Set the length from the tool holder to the tool tip.

---Setting range---

0.000 to 99999.999 (mm)

【#8044】 UNIT*10

Set the command increment scale.
 The scale will be "1" when "0" is set.

---Setting range---

0 to 10000 (fold)
 0: One fold

<3-dimensional tool radius compensation>

【#8071】 3-D CMP for M system only

Set the value of the denominator constants for 3-dimensional tool radius compensation.

Set the value of "p" in the following formula.

$$Vx = i \times r/p, Vy = j \times r/p, Vz = k \times r/p$$

Vx, Vy, Vz : X, Y, and Z axes or vectors of horizontal axes

i, j, k : Program command value

r : Offset

$p = \sqrt{(i^2 + j^2 + k^2)}$ when the set value is "0".

---Setting range---

0 to 99999.999

<Scaling>

【#8072】 SCALING P for M system only

Set the scale factor for reduction or magnification in the machining program specified by G50 or G51 command.

This parameter will be valid when the program specifies no scale factor.

---Setting range---

-99.999999 to 99.999999

<Spiral interpolation>

【#8075】 SpiralEndErr for M system only

Set the tolerable error range (absolute value) when the end point position, commanded by the spiral or conical interpolation command with the command format type 2, differs from the end point position obtained from the speed and increment/decrement amount.

---Setting range---

0 to 99999.999 (mm)

【#8078】 Screen Saver Timer

Set the period of time before turn-OFF of the display unit backlight.

When "0" is set, the backlight is not turned OFF.

---Setting range---

0 to 60 (min)

0: The backlight is not turned OFF

<Coord rotation>

【#8621】 Coord rot plane (H)

Set the plane (horizontal axis) for coordinate rotation control.

Usually, set the name of the 1st axis.

When not set, "X" axis will be set.

---Setting range---

Axis name

【#8622】 Coord rot plane (V)

Set the plane (vertical axis) for coordinate rotation control.

Usually, set the name of the 2nd axis.

When not set, "Y" axis will be set.

---Setting range---

Axis name

【#8623】 Coord rot centr (H)

Set the center coordinates (horizontal axis) for coordinate rotation control.

---Setting range---

-999999.999 to 999999.999 (mm)

[#8624] Coord rot centr (V)

Set the center coordinates (vertical axis) for coordinate rotation control.

---Setting range---

-999999.999 to 999999.999 (mm)

 [#8625] Coord rot vctr (H)

Set the vector components (horizontal axis) for coordinate rotation control.

When this parameter is set, the coordinate rotation control angle (#8627) will be automatically calculated.

---Setting range---

-999999.999 to 999999.999 (mm)

 [#8626] Coord rot vctr (V)

Set the vector components (vertical axis) for coordinate rotation control.

When this parameter is set, the coordinate rotation control angle (#8627) will be automatically calculated.

---Setting range---

-999999.999 to 999999.999 (mm)

 [#8627] Coord rot angle

Set the rotation angle for coordinate rotation control.

When this parameter is set, the coordinate rotation vector (#8625, #8626) will be "0".

---Setting range---

-360.000 to 360.000 (°)

<TLM> **[#8701] Tool length**

Set the length to the touch tool tip.

---Setting range---

±99999.999 (mm)

 [#8702] Tool Dia

Set the diameter of the sphere at the touch tool tip.

---Setting range---

±99999.999 (mm)

 [#8703] OFFSET X

This sets the deviation amount (X direction) from the touch tool center to the spindle center.

---Setting range---

±99999.999 (mm)

 [#8704] OFFSET Y

Set the deviation amount (Y direction) from the touch tool center to the spindle center.

---Setting range---

±99999.999 (mm)

 [#8705] RETURN

Set the one-time return distance for contacting again.

---Setting range---

0 to 99999.999 (mm)

 [#8706] FEED

Set the feedrate when contacting again.

---Setting range---

1 to 60000 (mm/min)

【#8707】 Skip past amount (H)

Set the difference (horizontal axis direction) between the skip read value and actual skip position.

---Setting range---
±99999.999 (mm)

【#8708】 Skip past amount (V)

Set the difference (vertical axis direction) between the skip read value and actual skip position.

---Setting range---
±99999.999 (mm)

【#8709】 EXT work sign rvs

Select when using the external workpiece coordinate system with Z shift.

Select whether to reverse the sign.

0: External workpiece offset (Z shift) without sign reversal

1: External workpiece offset (Z shift) with sign reversal

【#8710】 EXT work ofs invld

Set whether to enable external workpiece offset subtraction when setting the workpiece coordinate offset.

0: Not subtract the external workpiece offset. (Conventional specification)

1: Subtract the external workpiece offset.

【#8711】 TLM L meas axis

Set the tool length measurement axis.

Set the "#1022 axname2" axis name.

---Setting range---
Axis name
(Note) If the axis name is illegal or not set, the 3rd axis name will be set as default.

【#8712】 TLM D meas axis

Set the tool diameter measurement axis.

Set the "#1022 axname2" axis name.

---Setting range---
Axis name
(Note) If the axis name is illegal or not set, the 1st axis name will be set as default.

【#19001】 Syn.tap(,S)cancel

0: Retain the spindle speed (,S) in synchronous tap return

1: Cancel the spindle speed (,S) in synchronous tap return with G80

【#19002】 Zero-point mark

Select the position for displaying the zero point mark in the graphic trace and 2D check.

0: Machine coordinates zero point (same as conventional method)

1: Workpiece coordinate zero point

【#19003】 PRG coord rot type

Select the start point of the initial travel command after G68 command.

0: Calculate the end position using the current position on the local coordinate system before rotating, without rotating the start point in accordance with the coordinates rotation.

1: Calculate the end position, assuming that the start point rotates in accordance with the coordinates rotation.

<Surface speed control>

【#19425】 ManualB Std R1

Set a radius used as standard for the rotary axis speed.

When the setting value of #19425 is larger than that of "#19427 ManualB Std R2", #19425 setting will be used as surface speed control standard radius 2: #19427 setting will be used as surface speed control standard radius 1.

---Setting range---

0 to 99999.999 (mm)

【#19426】 ManualB Std F1

This sets the rotary axis speed for surface speed control standard radius 1 (ManualB Std R1).

When the setting value of #19426 is larger than that of "#19428 ManualB Std F2", #19426 setting will be used as surface speed control standard speed 2: #19427 setting will be used as surface speed control standard speed 1.

---Setting range---

1 to 1000000 (°/min)

【#19427】 ManualB Std R2

Set a radius used as standard for the rotary axis speed.

When the same value is set as "#19425 ManualB Std R1", the surface speed control standard speed 1 (ManualB Std F1) will be selected as the rotary axis speed if the radius is less than that value. The surface speed control standard speed 2 (ManualB Std F2) is selected if larger than the set value.

---Setting range---

0 to 99999.999 (mm)

【#19428】 ManualB Std F2

Set the rotary axis speed for surface speed control standard radius 2 (ManualB Std R2).

---Setting range---

1 to 1000000 (°/min)

9.2 Fixed Cycle

<FIXED C.> (Fixed cycle)

【#8012】	G73 n	for M system only
	Set the return amount for G73 (step cycle).	
	---Setting range---	
	0 to 99999.999 (mm)	
【#8013】	G83 n	
	Set the return amount for G83 (deep hole drilling cycle).	
	---Setting range---	
	0 to 99999.999 (mm)	
【#8014】	CDZ-VALE	for L system only
	Set the screw cut up amount for G76 and G78 (thread cutting cycle).	
	---Setting range---	
	0 to 127 (0.1 lead)	
【#8015】	CDZ-ANGLE	for L system only
	Set the screw cut up angle for G76 and G78 (thread cutting cycle).	
	---Setting range---	
	0 to 89 (°)	
【#8016】	G71 MINIMUM	for L system only
	Set the minimum value of the last cutting amount by the rough cutting cycle (G71, G72). The cutting amount of the last cutting will be the remainder. When the remainder is smaller than this parameter setting, the last cycle will not be executed.	
	---Setting range---	
	0 to 999.999 (mm)	
【#8017】	G71 DELTA-D	for L system only
	Set the change amount of the rough cutting cycle. The rough cutting cycle (G71, G72) cutting amount repeats $d+\Delta d$, d , $d-\Delta d$ using the value (d) commanded with D as a reference. Set the change amount Δd .	
	---Setting range---	
	0 to 999.999 (mm)	
【#8018】	G84/G74 n	for M system only
	Not used. Set to "0".	

<FIXED C.> (Fixed cycle)

【#8051】	G71 THICK
	Set the amount of cut-in by the rough cutting cycle (G71, G72)
	---Setting range---
	0 to 99999.999 (mm)
【#8052】	G71 PULL UP
	Set the amount of pull-up when returning to the cutting start point for the rough cutting cycle (G71, G72).
	---Setting range---
	0 to 99999.999 (mm)

【#8053】 G73 U

Set the X-axis cutting margin of the forming rough cutting cycle (G73).

---Setting range---

-99999.999 to 99999.999 (mm)

【#8054】 G73 W

Set the Z-axis cutting margin of the forming rough cutting cycle (G73).

---Setting range---

-99999.999 to 99999.999 (mm)

【#8055】 G73 R

Set how many times cutting will be performed in the forming rough cutting cycle (G73).

---Setting range---

0 to 99999 (times)

【#8056】 G74 RETRACT

Set the amount of retract (amount of cut-up) of the cutting-off cycle (G74, G75).

---Setting range---

0 to 999.999 (mm)

【#8057】 G76 LAST-D

Set the amount of final cut-in by the compound type thread cutting (G76).

---Setting range---

0 to 999.999 (mm)

【#8058】 G76 TIMES

Set how many times the amount of final cut-in cycle (G76 finish margin) will be divided in the compound type thread cutting (G76).

---Setting range---

0 to 99 (times)

【#8059】 G76 ANGLE

Set the angle (thread angle) of the tool nose in the compound type thread cutting (G76).

---Setting range---

0 to 99 (°)

<Deep hole drilling cycle>**【#8083】 G83S modeM for M system only**

Set the M command code for changing to the small diameter deep hole drilling cycle mode.

---Setting range---

1 to 99999999

【#8084】 G83S Clearance for M system only

Set the clearance amount for the small diameter deep hole drilling cycle (G83).

---Setting range---

0 to 999.999 (mm)

【#8085】 G83S Forward F for M system only

Set the feedrate from the R point to the cutting start position in the small diameter deep hole drilling cycle (G83).

---Setting range---

0 to 99999 (mm/min)

【#8086】 G83S Back F for M system only

Set the speed for returning from the hole bottom during the small diameter deep hole drilling cycle (G83).

---Setting range---

0 to 99999 (mm/min)

9.3 Control Parameters 1

【#1041(PR)】	I_inch	Initial inch
		Select the unit system for the program travel amount when the power is turned ON or reset and for position display. 0: Metric system 1: Inch system

(Note) Selection of inch and metric unit
When the setting value of "#1041 I_inch" is changed, the unit of length is changed after reset. The following parameters concerning length, however, are not changed automatically. Change the setting values of following parameters according to the new unit system.

Tool compensation amount (Tool length compensation amount, tool wear compensation amount and tool tip compensation amount)			
Workpiece coordinate offset			
Machining parameter	#8004 SPEED	#8027 Toler-1	#8056 G74 RETRACT
	#8005 ZONE r	#8028 Toler-2	#8057 G76 LAST-D
	#8006 ZONE d	#8029 FairingL	#8075 SpiralEndErr
	#8009 DSC. ZONE	#8030 MINUTE LENG	#8084 G83S Clearance
	#8010 ABS. MAX.	#8037 CorJudgeL	
	#8011 INC. MAX.	#8041 C-rot. R	#8085 G83S Forward F
	#8012 G73n	#8042 C-ins. R	#8086 G83S Back F
	#8013 G83n	#8051 G71 THICK	
	#8016 G71 MINIMUM	#8052 G71 PULL UP	
	#8017 G71 DELTA-D	#8053 G73 U	
Axis parameter	#8018 G84/G74n	#8054 G73 W	
	#8204 OT-CHECK-N		
	#8205 OT-CHECK-P		
	#8206 TOOL CHG.P		
Barrier data	#8209 G60 Shift		
	#8300-#8306, #8311-#8314		
Base specifications parameter	#1084 RadErr		

"#8004 SPEED" is 10 inches/min. unit for the inch system.

【#1078】	Decpt2	Decimal pnt type 2
		Select the increment of position commands that do not have a decimal point. 0: Minimum input command unit (follows "#1015 cunit") 1: 1mm (or 1inch) unit (For the dwell time, 1s unit is used.)

【#1080】	Dril_Z (For M system only)	Drilling Z fixed
		Select a fixed cycle hole drilling axis. 0: Use an axis vertical to the selected plane as hole drilling axis. 1: Use the Z axis as the hole drilling axis regardless of the selected plane.

【#1091】	Mpoint	Ignore middle pnt
		Select how to handle the middle point during G28 and G30 reference position return. 0: Pass the middle point designated in the program and move to the reference position. 1: Ignore the middle point designated in the program and move straight to the reference position.

【#1103】	T_life	T-life mgmt valid
		Select whether to use the tool life management. 0: Not use 1: Use

- 【#1104】 T_Com2 Tool cmd mthd 2**
 Select how to handle the tool command in the program when "#1103 T_Life" is set to "1".
 0: Handle the command as group No.
 1: Handle the command as tool No.
- 【#1105】 T_Sel2 Tool selection method 2**
 Select the tool selection method when "#1103 T_Life" is set to "1".
 0: Select in order of registered No. from the tools used in the same group.
 1: Select the tool with the longest remaining life from the tools used or unused in the same group.
- 【#1106】 Tcount Tool mgmt count for L system only**
 Select the input method when address N is omitted in inputting the data (G10 L3 command) for tool life management function II.
 0: Time specified input
 1: Number of times specified input
- 【#1126】 PB_G90 Playback G90**
 Select the method to command the playback travel amount in the playback editing.
 0: Incremental value
 1: Absolute value
- 【#1128】 RstVCI Com-var RET clear**
 Select how to handle the common variables when resetting.
 0: Common variables won't change after resetting.
 1: The following common variables will be cleared by resetting:
 #100 to #149 when 100 sets of variables are provided.
 #100 to #199 when 200 sets or more of variables are provided.
- 【#1129】 PwrVCI Clear variables by power-ON**
 Select how to handle the common variables when the power is turned ON.
 0: The common variables are in the same state as before turning the power OFF.
 1: The following common variables will be cleared when the power is turned ON:
 #100 to #149 when 100 sets of variables are provided.
 #100 to #199 when 200 sets or more of variables are provided.
- 【#1148】 I_G611 Initial hi-precis**
 Set the high accuracy control mode for the modal state when the power is turned ON.
 0: G64 (cutting mode) at power ON
 1: G61.1 (high-accuracy control mode) at power ON
- 【#1302】 AutoRP Auto restart valid**
 Select the method to move to the restart position when restarting the program.
 0: Move the system manually to the restart position and then restart the program.
 1: The system automatically moves to the restart position at the first activation after the program restarts.
- 【#8101】 MACRO SINGLE**
 Select how to control the blocks where the user macro command continues.
 0: Do not stop while macro blocks continue.
 1: Stop every block during signal block operation.
- 【#8102】 COLL. ALM OFF**
 Select the interference (bite) control to the workpiece from the tool diameter during tool radius compensation and nose R compensation.
 0: An alarm will be output and operation stops when an interference is judged.
 1: Changes the path to avoid interference.

【#8103】 COLL. CHK OFF

Select the interference (bite) control to the workpiece from the tool diameter during tool radius compensation and nose R compensation.

- 0: Performs interference check.
- 1: Does not perform interference check.

【#8105】 EDIT LOCK B

Select the edit lock for program Nos. 8000 to 9999 in the memory.

- 0: Enable the editing.
- 1: Prohibit the editing of above programs.

When "1" is set, the file cannot be opened.

【#8106】 G46 NO REV-ERR for L system only

Select the control for the compensation direction reversal in G46 (nose R compensation).

- 0: An alarm will be output and operation will stop when the compensation direction is reversed (G41 -> G42' G42 -> G41).
- 1: An alarm won't occur when the compensation direction is reversed, and the current compensation direction will be maintained.

【#8107】 R COMPENSATION

Select whether to move to the inside because of a delay in servo response to a command during arc cutting mode.

- 0: Move to the inside, making the arc smaller than the command value.
- 1: Compensate the movement to the inside.

【#8108】 R COMP Select

Select the arc radius error compensation target.

- 0: Perform compensation over all axes.
- 1: Perform compensation axis by axis.

(Note) This parameter is effective only when "#8107 R COMPENSATION" is "1".

【#8109】 HOST LINK

Select whether to enable computer link B instead of the RS-232C port.

- 0: Disable (Enable normal RS-232C communication.)
- 1: Enable (Disable normal RS-232C communication.)

【#8110】 G71/G72 POCKET

Select whether to enable the pocket machining when there is a dimple (pocket) in the rough cutting cycle (G71, G72) finishing program.

- 0: OFF
- 1: ON

【#8111】 Milling Radius

Select the diameter and radius of the linear axis for milling (cylindrical/pole coordinate) interpolation.

- 0: All axes radius command
- 1: Each axis setting (follows "#1019 dia")

(Note) This parameter is valid only in the milling (cylindrical/polar coordinate) interpolation mode.

【#8112】 DECIMAL PNT-P

Select whether to enable the decimal point command for G04 address P.

- 0: Disable
- 1: Enable

【#8113】 Milling Init G16

Set which plane to execute for milling machining after the power is turned ON or reset.

#8113	#8114	Plane
0	0	G17 plane
0	1	G19 plane
1	0	G16 plane
1	1	

0: Not G16 plane

1: G16 plane

(Note) This parameter is valid for the G code system 2 or 3 ("#1037 cmdtyp"="3" or "4").

【#8114】 Milling Init G19

Set which plane to execute for milling machining after the power is turned ON or reset.

#8113	#8114	Plane
0	0	G17 plane
0	1	G19 plane
1	0	G16 plane
1	1	

0: Not G19 plane

1: G19 plane

(Note) This parameter is valid for the G code system 2 or 3 ("#1037 cmdtyp"="3" or "4").

【#8116】 Coord rot para invd

Select whether to enable the coordinate rotation by the parameters.

0: Enable

1: Disable

【#8117】 OFS Diam DESIGN

Select tool radius or tool diameter compensation amount to be specified.

0: Tool radius compensation amount

1: Tool diameter compensation amount

【#8119】 Comp. unit switch

Select the setting unit of compensation amount that has no decimal point.

0: 1mm (or 1inch) unit

1: The minimum command unit (follows "#1003 iunit")

【#8121】 Screen Capture

Select whether to enable the screen capture function.

0: Disable

1: Enable

(Note1) By setting this parameter to "1", and by keeping pushing the [SHIFT] key, screen capture will be executed.

(Note2) This parameter is valid only with M70 Series.

【#8122】 Keep G43 MDL M-REF

Select whether to keep the tool length offset by high speed manual reference position return during tool length offset.

0: Will not be kept (Cancel)

1: Kept

【#8124】 Mirr img at reset

Select the operation type of the mirror image by parameter setting and the mirror image by external input.

0: The current mirror image is canceled, and new mirror image will start with the machine position at reset as the mirror center.

1: The mirror center is kept to continue the mirror image.

【#8145】 Validate F1 digit

Select whether to execute the F command with a 1-digit code command or with a direct numerical command.

0: Direct numerical command (command feedrate during feed per minute or rotation)

1: 1-digit code command (with the feedrate specified by the parameters "#1185 spd_F1" to "#1189 F5")

【#8154(PR)】

Not used. Set to "0".

【#8155】 Sub-pro interrupt

Select the method for the user macro interrupt.

0: The user macro interrupt of macro type

1: The user macro interrupt of sub-program type

【#8156】 Fine thread cut E

Select the address E type when cutting an inch screw.

0: Specify the number of threads per inch for inch screw cutting.

1: Specify the precision lead for inch screw cutting.

【#8157】 Radius comp type B (M system) / Nose R comp type B (L system)

For M system

Select the method of the arithmetic processing for the intersection point when the start-up or cancel commands are operated during radius compensation.

0: The processing does not handle the start-up or cancel command block: handle the offset vector in the direction vertical to that of the command instead.

1: The processing is executed for the intersection point between the command block and the next block.

For L system

Select the method of the arithmetic processing for the intersection point when the start-up or cancel commands are operated during nose R or radius compensation.

0: The processing does not handle the start-up or cancel command block: handle the offset vector in the direction vertical to that of the command instead.

1: The processing is executed for the intersection point between the command block and the next block.

【#8158】 Init const sur spd

Select the initial state after power-ON.

0: Constant surface speed control cancel mode.

1: Constant surface speed control mode.

【#8159】 Synchronous tap

Select whether to use the floating tap chuck in G74 and G84 tap cycles.

0: With a floating tapping chuck

1: Without a floating tapping chuck

【#8160】 Start point alarm

Select an operation when the operation start point cannot be found while moving to the next block of G117.

0: The auxiliary function is enabled after the block for the movement has finished.

1: The program error (P33) occurs.

9.4 Control Parameters 2

【#1025】 I_plane Initial plane

Select the plane to be selected when the power is turned ON or reset.

- 0: X-Y plane (G17 command state)
- 1: X-Y plane (G17 command state)
- 2: Z-X plane (G18 command state)
- 3: Y-Z plane (G19 command state)

【#1037(PR)】 cmdtyp Command type

Set the G code list and compensation type for programs.

cmdtyp	G code list	Compensation type
1	List 1 (for M)	Type A (one compensation amount for one compensation No.)
2	List 1 (for M)	Type B (shape and wear compensation amounts for one compensation No.)
3	List 2 (for L)	Type C (shape and wear compensation amounts for one compensation No.)
4	List 3 (for L)	Ditto
5	List 4 (for special L)	Ditto
6	List 5 (for special L)	Ditto
7	List 6 (for special L)	Ditto
8	List 7 (for special L)	Ditto
9	List 8 (for M) M2 format type A	Type A (one compensation amount for one compensation No.)
10	List 8 (for M) M2 format type B	Type B (shape and wear amounts for one compensation No.)

There are some items in the specifications that can be used or cannot be used according to the value set in this parameter.

The file structure may also change depending on the compensation data type.

【#1073】 I_Absm Initial absolute

Select the mode (absolute or incremental) at turning ON the power or reset.

- 0: Incremental setting
- 1: Absolute setting

【#1074】 I_Sync Initial sync feed

Select the feedrate mode at turning ON the power or reset.

- 0: Asynchronous feed (feed per minute)
- 1: Synchronous feed (feed per revolution)

【#1075】 I_G00 Initial G00

Select the linear command mode at turning ON the power or reset.

- 0: Linear interpolation (G01 command state)
- 1: Positioning (G00 command state)

【#1076】 AbsInc ABS/INC address for L system only

Select the command method for the absolute and incremental commands.

- 0: Use G command for the absolute and incremental commands.
- 1: Use axis name for the absolute and incremental commands.

(The axis name in "#1013 axname" will be the absolute command, "#1014 incax" will be the incremental command.)

When "1" is selected, using two axis names, one each for the absolute and incremental commands, allows to issue the absolute and incremental commands appropriately to an axis.

【#1085】 G00Drn G00 dry run

Select whether to apply dry run (feed at manual setting speed instead of command feedrate) to the G00 command.

0: Not apply to G00. (move at rapid traverse rate)

1: Apply to G00. (move at manual setting speed)

【#1086】 G0Intp G00 interp OFF

Select the G00 travel path type.

0: Move linearly toward the end point. (interpolation type)

1: Move to the end point of each axis at the rapid traverse feedrate for each axis. (non-interpolation)

(Note) If this parameter is set to "1", neither of the following functions will be available: rapid traverse constant inclination acceleration/deceleration and rapid traverse constant inclination multi-step acceleration/deceleration.

9.5 I/O Parameters

There are basically two types of input/output parameters which must be set when inputting, outputting or referring to data, or when performing tape operation.

9001 to 9018, 9051, 9052 parameters:

Set which channel to connect which device to for each I/O application.

901 to 9528 parameters:

Set the transmission speed, etc., for each input/output device.

Up to five types of input/output device parameters can be set in device 0 to 4.

【#9001】 DATA IN PORT

Select the port for inputting the data such as machine program and parameters.

- 1: ch1
- 2: ch2

【#9002】 DATA IN DEV.

Select the device No. for inputting the data.

(The device Nos. correspond to the input/output device parameters.)

---Setting range---

0 to 4

【#9003】 DATA OUT PORT

Select the port for outputting the data such as machine program and parameters.

- 1: ch1
- 2: ch2

【#9004】 DATA OUT DEV.

Select the device No. for outputting the data.

(The device Nos. correspond to the input/output device parameters.)

---Setting range---

0 to 4

【#9005】 TAPE MODE PORT

Select the input port for running with the tape mode.

- 1: ch1
- 2: ch2

【#9006】 TAPE MODE DEV.

Select the device No. to be run with the tape mode.

(The device Nos. correspond to the input/output device parameters.)

---Setting range---

0 to 4

【#9007】 MACRO PRINT PORT

Select the output port used for the user macro DPRINT command.

- 1: ch1
- 2: ch2

【#9008】 MACRO PRINT DEV.

Select the device No. used for the DPRINT command.

(The device Nos. correspond to the input/output device parameters.)

---Setting range---

0 to 4

【#9009】 PLC IN/OUT PORT

Select the port for inputting/outputting various data with PLC.

- 1: ch1
- 2: ch2

【#9010】 PLC IN/OUT DEV.

Select the device No. used for the PLC input/output.
(The device Nos. correspond to the input/output device parameters.)

- Setting range---
0 to 4

【#9011】 REMOTE PRG IN PORT

Select the port for inputting remote programs.

- 1: ch1
- 2: ch2

【#9012】 REMOTE PRG IN DEV.

Select the device No. used to input remote programs.
The device Nos. correspond to the input/output device parameters.

- Setting range---
0 to 4

【#9013】 EXT UNIT PORT

Select the port for communication with an external unit.

- 1: ch1
- 2: ch2

【#9014】 EXT UNIT DEV.

Select the unit No. used for communication with an external unit
(The unit Nos. correspond to the input/output device parameters.)

- Setting range---
0 to 4

【#9017】 HANDY TERMINAL PORT

Select the port for communication with a handy terminal.

- 1: ch1
- 2: ch2

【#9018】 HANDY TERMINAL DEV.

Select the device No. used for communication with a handy terminal.
(The device Nos. correspond to the input/output device parameters.)

- Setting range---
0 to 4

【#9051】 Data I/O port

Select whether to use display side serial port or NC side serial port for data input/output function.

- 0: Display side serial port
- 1: Display side serial port
- 2: NC side serial port

(Note) The setting range differs according to the model.

【#9052】 Tape mode port

Select whether to use display side serial port or NC side serial port for tape mode.

- 0: NC side serial port
- 1: Display side serial port
- 2: NC side serial port

(Note) The setting range differs according to the model.

【#9101】 DEVO DEVICE NAME

Set the device name corresponding to the device No.
Set a simple name for quick identification.

---Setting range---

Use alphabet characters, numerals and symbols to set a name within 3 characters.

【#9102】 DEVO BAUD RATE

Select the serial communication speed.

- 0: 19200 (bps)
- 1: 9600
- 2: 4800
- 3: 2400
- 4: 1200
- 5: 600
- 6: 300
- 7: 110

【#9103】 DEVO STOP BIT

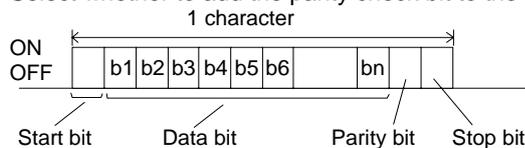
Select the stop bit length used in the start-stop system.

Refer to "#9104 DEVO PARITY CHECK". At the output of data, the number of characters is always adjusted for the parity check.

- 1: 1 (bit)
- 2: 1.5
- 3: 2

【#9104】 DEVO PARITY CHECK

Select whether to add the parity check bit to the data.



Set this parameter in accordance with the I/O device specifications.

- 0: Not add a parity bit in I/O mode
- 1: Add a parity bit in I/O mode

【#9105】 DEVO EVEN PARITY

Select odd or even when parity is added to the data. This parameter is ignored when no parity is added.

- 0: Odd parity
- 1: Even parity

【#9106】 DEVO CHR. LENGTH

Set the length of the data bit.

Refer to "#9104 DEVO PARITY CHECK".

- 0: 5 (bit)
- 1: 6
- 2: 7 (NC connection not supported)
- 3: 8

【#9107】 DEVO TERMINATR TYP

Select the code to terminate data reading.

- 0, 3: EOR
- 1, 2: EOB or EOR

[#9108] DEVO HAND SHAKE

Select the transmission control method.
 No handshaking will be used when a value except 1 to 3 is set.
 1: RTS/CTS method
 2: No handshaking
 3: DC code method

 [#9109] DEVO DC CODE PRY

Select the DC code type when the DC code method is selected.
 0: Not add parity to DC code (DC3 = 13H)
 1: Add parity to DC code (DC3 = 93H)

 [#9111] DEVO DC2/4 OUTPUT

Select the DC code handling when outputting data to the output device.

---Setting range---
 DC2 / DC4
 0: None / None
 1: Yes / None
 2: None / Yes
 3: Yes / Yes

 [#9112] DEVO CR OUTPUT

Select whether to add the (CR) code just before the EOB (L/F) code during output.
 0: Not add
 1: Add

 [#9113] DEVO EIA OUTPUT

Select ISO or EIA code for data output.
 In data input mode, the ISO and EIA codes are identified automatically.
 0: ISO code output
 1: EIA code output

 [#9114] DEVO FEED CHR.

Set the length of the tape feed to be output at the start and end of the data during tape output.

---Setting range---
 0 to 999 (characters)

 [#9115] DEVO PARITY V

Select whether to perform the parity check for the number of characters in a block at the input of data.
 At the output of data, the number of characters is always adjusted to for the parity check.
 0: Not perform parity V check
 1: Perform parity V check

 [#9116] DEVO TIME-OUT (sec)

Set the time out time to detect an interruption in communication.
 Time out check will not be executed when set to "0".

---Setting range---
 0 to 30 (s)

 [#9117] DEVO DR OFF

Select whether to enable the DR data check in data I/O mode.
 0: Enable
 1: Disable

 [#9118] DEVO DATA ASCII

Select the code of the output data.
 0: ISO/EIA code
 (Depends on whether #9113, #9213, #9313, #9413 or #9513 EIA output parameter is set up.)
 1: ASCII code

[#9119] DEV0 INPUT TYPE

Select the mode for input (verification).

0: Standard input (Data from the very first EOB is handled as significant information.)

1: EOBs following the first EOB of the input data are skipped until data other than EOB is input

 [#9121] DEV0 EIA CODE [

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code "[" .

When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

---Setting range---

0 to FF (hexadecimal)

 [#9122] DEV0 EIA CODE]

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code "]" .

When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

---Setting range---

0 to FF (hexadecimal)

 [#9123] DEV0 EIA CODE #

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code "#" .

When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

---Setting range---

0 to FF (hexadecimal)

 [#9124] DEV0 EIA CODE *

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code "*" .

When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

---Setting range---

0 to FF (hexadecimal)

 [#9125] DEV0 EIA CODE =

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code "=" .

When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

---Setting range---

0 to FF (hexadecimal)

 [#9126] DEV0 EIA CODE :

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code ":" .

When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

---Setting range---

0 to FF (hexadecimal)

【#9127】 DEV0 EIA CODE \$

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code "\$".

When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

---Setting range---

0 to FF (hexadecimal)

【#9128】 DEV0 EIA CODE !

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code "!".

When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

---Setting range---

0 to FF (hexadecimal)

【#9201】 DEV1 DEVICE NAME

Set the device name corresponding to the device No.

Set a simple name for quick identification.

<Setting range>

Use alphabet characters, numerals and symbols to set a name within 3 characters.

【#9202】 DEV1 BAUD RATE

Select the serial communication speed.

0: 19200 (bps)

1: 9600

2: 4800

3: 2400

4: 1200

5: 600

6: 300

7: 110

【#9203】 DEV1 STOP BIT

Select the stop bit length used in the start-stop system.

Refer to "#9204 DEV1 PARITY CHECK". At the output of data, the number of characters is always adjusted to for the parity check.

<Setting range>

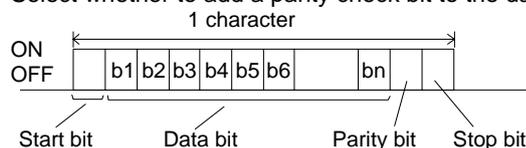
1: 1 (bit)

2: 1.5

3: 2

【#9204】 DEV1 PARITY CHECK

Select whether to add a parity check bit to the data.



Set this parameter in accordance with the I/O device specifications.

0: Not add a parity bit in I/O mode

1: Add a parity bit in I/O mode

【#9205】 DEV1 EVEN PARITY

Select whether even or odd parity will be used when parity is used. This parameter is ignored when no parity is added.

0: Odd parity

1: Even parity

【#9206】 DEV1 CHR. LENGTH

Select the length of the data bit.
Refer to "#9204 DEV1 PARITY CHECK".
<Setting range>
0: 5 (bit)
1: 6
2: 7 (NC connection not supported)
3: 8

【#9207】 DEV1 TERMINATR TYP

Select the code to terminate data reading.
0,3: EOR
1,2: EOB or EOR

【#9208】 DEV1 HAND SHAKE

Select the transmission control method.
No handshaking will be used when a value except 1 to 3 is set.
1: RTS/CTS method
2: No handshaking
3: DC code method

【#9209】 DEV1 DC CODE PRTY

Select the DC code type when the DC code method is selected.
0: Not add parity to DC code (DC3 = 13H)
1: Add parity to DC code (DC3 = 93H)

【#9211】 DEV1 DC2/4 OUTPUT

Select the DC code handling when outputting data to the output device.

<Setting range>
DC2 / DC4
0: None / None
1: Yes / None
2: None / Yes
3: Yes / Yes

【#9212】 DEV1 CR OUTPUT

Select whether to add the (CR) code just before the EOB (L/F) code during output.
0: Not add
1: Add

【#9213】 DEV1 EIA OUTPUT

Select ISO or EIA code for data output.
In data input mode, the ISO and EIA codes are identified automatically.
0: ISO code output
1: EIA code output

【#9214】 DEV1 FEED CHR.

Set the length of the tape feed to be output at the start and end of the data during tape output.

<Setting range>
0 to 999 (characters)

【#9215】 DEV1 PARITY V

Select whether to perform the parity check for the number of characters in a block at the input of data.
At the output of data, the number of characters is always adjusted to for the parity check.
0: Not perform parity V check
1: Perform parity V check

【#9216】 DEV1 TIME-OUT (sec)

Set the time out time to detect an interruption in communication.
Time out check will not be executed when set to "0".

<Setting range>
0 to 30 (s)

【#9217】 DEV1 DR OFF

Select whether to enable the DR data check in data I/O mode.
0: Enable
1: Disable

【#9218】 DEV1 DATA ASCII

Select the code of the output data.
0: ISO/EIA code
(Depends on whether #9113, #9213, #9313, #9413 or #9513 EIA output parameter is set up.)
1: ASCII code

【#9219】 DEV1 INPUT TYPE

Select the mode for input (verification).
0: Standard input (Data from the very first EOB is handled as significant information.)
1: EOBs following the first EOB of the input data are skipped until data other than EOB is input

【#9221】 DEV1 EIA CODE [

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code "[".
When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

<Setting range>
0 to FF (hexadecimal)

【#9222】 DEV1 EIA CODE]

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code "]"
".
When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

<Setting range>
0 to FF (hexadecimal)

【#9223】 DEV1 EIA CODE #

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code "#".
When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified. .

<Setting range>
0 to FF (hexadecimal)

【#9224】 DEV1 EIA CODE *

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code "*".
When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

<Setting range>
0 to FF (hexadecimal)

【#9225】 DEV1 EIA CODE =

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code "=".

When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

<Setting range>
0 to FF (hexadecimal)

【#9226】 DEV1 EIA CODE :

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code ":".

When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

<Setting range>
0 to FF (hexadecimal)

【#9227】 DEV1 EIA CODE \$

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code "\$".

When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

<Setting range>
0 to FF (hexadecimal)

【#9228】 DEV1 EIA CODE !

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code "!".

When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

<Setting range>
0 to FF (hexadecimal)

【#9301】 DEV2 DEVICE NAME

Set the device name corresponding to the device No.
Set a simple name for quick identification.

<Setting range>
Use alphabet characters, numerals and symbols to set a name within 3 characters.

【#9302】 DEV2 BAUD RATE

Select the serial communication speed.

0: 19200 (bps)
1: 9600
2: 4800
3: 2400
4: 1200
5: 600
6: 300
7: 110

【#9303】 DEV2 STOP BIT

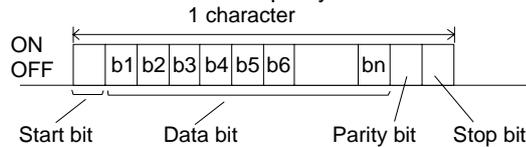
Select the stop bit length used in the start-stop system.

Refer to "#9304 DEV2 PARITY CHECK". At the output of data, the number of characters is always adjusted to for the parity check.

1: 1 (bit)
2: 1.5
3: 2

【#9304】 DEV2 PARITY CHECK

Select whether to add a parity check bit to the data.



Set this parameter in accordance with the I/O device specifications.

0: Not add a parity bit in I/O mode

1: Add a parity bit in I/O mode

【#9305】 DEV2 EVEN PARITY

Select whether even or odd parity will be used when parity is used. This parameter is ignored when no parity is added.

0: Odd parity

1: Even parity

【#9306】 DEV2 CHR. LENGTH

Select the length of the data bit.

Refer to "#9304 DEV2 PARITY CHECK".

0: 5 (bit)

1: 6

2: 7 (NC connection not supported)

3: 8

【#9307】 DEV2 TERMINATR TYP

Select the code to terminate data reading.

0, 3: EOR

1, 2: EOB or EOR

【#9308】 DEV2 HAND SHAKE

Select the transmission control method.

No handshaking will be used when a value except 1 to 3 is set.

1: RTS/CTS method

2: No handshaking

3: DC code method

【#9309】 DEV2 DC CODE PRTY

Select the DC code type when the DC code method is selected.

0: Not add parity to DC code (DC3 = 13H)

1: Add parity to DC code (DC3 = 93H)

【#9311】 DEV2 DC2/4 OUTPUT

Select the DC code handling when outputting data to the output device.

<Setting range>

DC2 / DC4

0: None / None

1: Yes / None

2: None / Yes

3: Yes / Yes

【#9312】 DEV2 CR OUTPUT

Select whether to add the (CR) code just before the EOB (L/F) code during output.

0: Not add

1: Add

【#9313】 DEV2 EIA OUTPUT

Select ISO or EIA code for data output.

In data input mode, the ISO and EIA codes are identified automatically.

0: ISO code output

1: EIA code output

[#9314] DEV2 FEED CHR.

Set the length of the tape feed to be output at the start and end of the data during tape output.

<Setting range>
0 to 999 (characters)

 [#9315] DEV2 PARITY V

Select whether to perform the parity check for the number of characters in a block at the input of data. At the output of data, the number of characters is always adjusted to for the parity check.

0: Not perform parity V check
1: Perform parity V check

 [#9316] DEV2 TIME-OUT (sec)

Set the time out time to detect an interruption in communication. Time out check will not be executed when set to "0".

<Setting range>
0 to 30 (s)

 [#9317] DEV2 DR OFF

Select whether to enable the DR data check in data I/O mode.

0: Enable
1: Disable

 [#9318] DEV2 DATA ASCII

Select the code of the output data.

0: ISO/EIA code
(Depends on whether #9113, #9213, #9313, #9413 or #9513 EIA output parameter is set up.)
1: ASCII code

 [#9319] DEV2 INPUT TYPE

Select the mode for input (verification).

0: Standard input (Data from the very first EOB is handled as significant information.)
1: EOBs following the first EOB of the input data are skipped until data other than EOB is input

 [#9321] DEV2 EIA CODE [

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code "[".

When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

<Setting range>
0 to FF (hexadecimal)

 [#9322] DEV2 EIA CODE]

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code "]"

When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

<Setting range>
0 to FF (hexadecimal)

 [#9323] DEV2 EIA CODE #

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code "#".

When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

<Setting range>
0 to FF (hexadecimal)

【#9324】 DEV2 EIA CODE *

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code "*".

When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

<Setting range>
0 to FF (hexadecimal)

【#9325】 DEV2 EIA CODE =

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code "=".

When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

<Setting range>
0 to FF (hexadecimal)

【#9326】 DEV2 EIA CODE :

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code ":".

When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

<Setting range>
0 to FF (hexadecimal)

【#9327】 DEV2 EIA CODE \$

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code "\$".

When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

<Setting range>
0 to FF (hexadecimal)

【#9328】 DEV2 EIA CODE !

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code "!".

When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

<Setting range>
0 to FF (hexadecimal)

【#9401】 DEV3 DEVICE NAME

Set the device name corresponding to the device No.
Set a simple name for quick identification.

<Setting range>
Use alphabet characters, numerals and symbols to set a name within 3 characters.

【#9402】 DEV3 BAUD RATE

Select the serial communication speed.

0: 19200 (bps)
1: 9600
2: 4800
3: 2400
4: 1200
5: 600
6: 300
7: 110

【#9403】 DEV3 STOP BIT

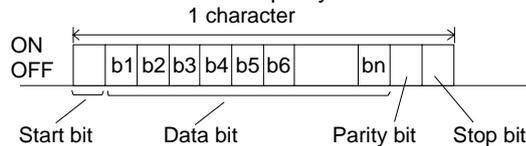
Select the stop bit length used in the start-stop system.

Refer to "#9404 DEV3 PARITY CHECK". At the output of data, the number of characters is always adjusted to for the parity check.

- 1: 1 (bit)
- 2: 1.5
- 3: 2

【#9404】 DEV3 PARITY CHECK

Select whether to add a parity check bit to the data.



Set this parameter in accordance with the I/O device specifications.

- 0: Not add a parity bit in I/O mode
- 1: Add a parity bit in I/O mode

【#9405】 DEV3 EVEN PARITY

Select whether even or odd parity will be used when parity is used. This parameter is ignored when no parity is added.

- 0: Odd parity
- 1: Even parity

【#9406】 DEV3 CHR. LENGTH

Select the length of the data bit.

Refer to "#9404 DEV3 PARITY CHECK".

- 0: 5 (bit)
- 1: 6
- 2: 7 (NC connection not supported)
- 3: 8

【#9407】 DEV3 TERMINATR TYP

Select the code to terminate data reading.

- 0, 3: EOR
- 1, 2: EOB or EOR

【#9408】 DEV3 HAND SHAKE

Select the transmission control method.

No handshaking will be used when a value except 1 to 3 is set.

- 1: RTS/CTS method
- 2: No handshaking
- 3: DC code method

【#9409】 DEV3 DC CODE PRTY

Select the DC code type when the DC code method is selected.

- 0: Not add parity to DC code (DC3 = 13H)
- 1: Add parity to DC code (DC3 = 93H)

【#9411】 DEV3 DC2/4 OUTPUT

Select the DC code handling when outputting data to the output device.

<Setting range>

DC2 / DC4

- 0: None / None
- 1: Yes / None
- 2: None / Yes
- 3: Yes / Yes

【#9412】 DEV3 CR OUTPUT

Select whether to add the (CR) code just before the EOB (L/F) code during output.

- 0: Not add
- 1: Add

【#9413】 DEV3 EIA OUTPUT

Select ISO or EIA code for data output.

In data input mode, the ISO and EIA codes are identified automatically.

- 0: ISO code output
- 1: EIA code output

【#9414】 DEV3 FEED CHR.

Set the length of the tape feed to be output at the start and end of the data during tape output.

- <Setting range>
0 to 999 (characters)

【#9415】 DEV3 PARITY V

Select whether to perform the parity check for the number of characters in a block at the input of data.

At the output of data, the number of characters is always adjusted to for the parity check.

- 0: Not perform parity V check
- 1: Perform parity V check

【#9416】 DEV3 TIME-OUT (sec)

Set the time out time to detect an interruption in communication.

Time out check will not be executed when set to "0".

- <Setting range>
0 to 30 (s)

【#9417】 DEV3 DR OFF

Select whether to enable the DR data check in data I/O mode.

- 0: Enable
- 1: Disable

【#9418】 DEV3 DATA ASCII

Select the code of the output data.

0: ISO/EIA code

(Depends on whether #9113, #9213, #9313, #9413 or #9513 EIA output parameter is set up.)

- 1: ASCII code

【#9419】 DEV3 INPUT TYPE

Select the mode for input (verification).

0: Standard input (Data from the very first EOB is handled as significant information.)

1: EOBs following the first EOB of the input data are skipped until data other than EOB is input.

【#9421】 DEV3 EIA CODE [

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code "[".

When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

- <Setting range>
0 to FF (hexadecimal)

【#9422】 DEV3 EIA CODE]

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code "]"
".
When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

<Setting range>
0 to FF (hexadecimal)

【#9423】 DEV3 EIA CODE #

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code "#".
When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

<Setting range>
0 to FF (hexadecimal)

【#9424】 DEV3 EIA CODE *

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code "*".
When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

<Setting range>
0 to FF (hexadecimal)

【#9425】 DEV3 EIA CODE =

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code "=".
When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

<Setting range>
0 to FF (hexadecimal)

【#9426】 DEV3 EIA CODE :

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code ":"
".
When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

<Setting range>
0 to FF (hexadecimal)

【#9427】 DEV3 EIA CODE \$

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code "\$".
When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

<Setting range>
0 to FF (hexadecimal)

【#9428】 DEV3 EIA CODE !

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code "!".
When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

<Setting range>
0 to FF (hexadecimal)

【#9501】 DEV4 DEVICE NAME

Set the device name corresponding to the device No.
Set a simple name for quick identification.

<Setting range>

Use alphabet characters, numerals and symbols to set a name within 3 characters.

【#9502】 DEV4 BAUD RATE

Select the serial communication speed.

- 0: 19200 (bps)
- 1: 9600
- 2: 4800
- 3: 2400
- 4: 1200
- 5: 600
- 6: 300
- 7: 110

【#9503】 DEV4 STOP BIT

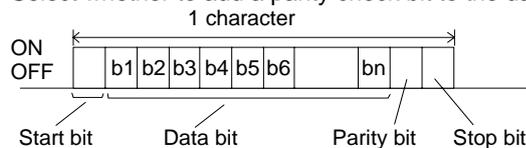
Select the stop bit length used in the start-stop system.

Refer to "#9504 DEV4 PARITY CHECK". At the output of data, the number of characters is always adjusted to for the parity check.

- 1: 1 (bit)
- 2: 1.5
- 3: 2

【#9504】 DEV4 PARITY CHECK

Select whether to add a parity check bit to the data.



Set this parameter in accordance with the I/O device specifications.

- 0: Not add a parity bit in I/O mode
- 1: Add a parity bit in I/O mode

【#9505】 DEV4 EVEN PARITY

Select whether even or odd parity will be used when parity is used. This parameter is ignored when no parity is added.

- 0: Odd parity
- 1: Even parity

【#9506】 DEV4 CHR. LENGTH

Select the length of the data bit.

Refer to "#9504 DEV4 PARITY CHECK".

- 0: 5 (bit)
- 1: 6
- 2: 7 (NC connection not supported)
- 3: 8

【#9507】 DEV4 TERMINATR TYP

Select the code to terminate data reading.

- 0, 3: EOR
- 1, 2: EOB or EOR

【#9508】 DEV4 HAND SHAKE

Select the transmission control method.

No handshaking will be used when a value except 1 to 3 is set.

- 1: RTS/CTS method
- 2: No handshaking
- 3: DC code method

【#9509】 DEV4 DC CODE PRTY

Select the DC code type when the DC code method is selected.
 0: Not add parity to DC code (DC3 = 13H)
 1: Add parity to DC code (DC3 = 93H)

【#9511】 DEV4 DC2/4 OUTPUT

Select the DC code handling when outputting data to the output device.

<Setting range>
 DC2 / DC4
 0: None / None
 1: Yes / None
 2: None / Yes
 3: Yes / Yes

【#9512】 DEV4 CR OUTPUT

Select whether to add the (CR) code just before the EOB (L/F) code during output.
 0: Not add
 1: Add

【#9513】 DEV4 EIA OUTPUT

Select ISO or EIA code for data output.
 In data input mode, the ISO and EIA codes are identified automatically.
 0: ISO code output
 1: EIA code output

【#9514】 DEV4 FEED CHR.

Set the length of the tape feed to be output at the start and end of the data during tape output.

<Setting range>
 0 to 999 (characters)

【#9515】 DEV4 PARITY V

Select whether to perform the parity check for the number of characters in a block at the input of data.
 At the output of data, the number of characters is always adjusted to for the parity check.
 0: Not perform parity V check
 1: Perform parity V check

【#9516】 DEV4 TIME-OUT (sec)

Set the time out time to detect an interruption in communication.
 Time out check will not be executed when set to "0".

<Setting range>
 0 to 30 (s)

【#9517】 DEV4 DR OFF

Select whether to enable the DR data check in data I/O mode.
 0: Enable
 1: Disable

【#9518】 DEV4 DATA ASCII

Select the code of the output data.
 0: ISO/EIA code
 (Depends on whether #9113, #9213, #9313, #9413 or #9513 EIA output parameter is set up.)
 1: ASCII code

【#9519】 DEV4 INPUT TYPE

Select the mode for input (verification).
 0: Standard input (Data from the very first EOB is handled as significant information.)
 1: EOBs following the first EOB of the input data are skipped until data other than EOB is input.

【#9521】 DEV4 EIA CODE [

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code "[".
When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

<Setting range>
0 to FF (hexadecimal)

【#9522】 DEV4 EIA CODE]

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code "]".
When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

<Setting range>
0 to FF (hexadecimal)

【#9523】 DEV4 EIA CODE #

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code "#".
When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

<Setting range>
0 to FF (hexadecimal)

【#9524】 DEV4 EIA CODE *

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code "*".
When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

<Setting range>
0 to FF (hexadecimal)

【#9525】 DEV4 EIA CODE =

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code "=".
When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

<Setting range>
0 to FF (hexadecimal)

【#9526】 DEV4 EIA CODE :

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code ":".
When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

<Setting range>
0 to FF (hexadecimal)

【#9527】 DEV4 EIA CODE \$

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code "\$".
When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

<Setting range>
0 to FF (hexadecimal)

【#9528】 DEV4 EIA CODE !

Set the code in hexadecimal, which does not duplicate the existing EIA codes, for the special code "!".

When output with EIA code, data can be output using the alternate code in which the special ISO code, not included in EIA, is specified.

<Setting range>

0 to FF (hexadecimal)

9.6 Ethernet Parameters

Set the parameters related to Ethernet input/output.

After setting up the parameter (PR) listed in the table, turn OFF the NC power. To validate the parameter, turn ON the power again.

When the Ethernet parameter setting is mistaken, NC screen might not be able to be displayed because the communication between the display unit and the control unit is disabled.

When the communication between the display unit and the control unit has been disabled, the connectable control unit IP address list screen is displayed at NC startup.

Reset the Ethernet parameter according to the "Appendix 8. IP Address Resetting Procedure at Disabled Network Communication [700 Series Only]".

11005,9701,9706 parameters:

When several TCP/IP drivers are installed and the IP address is set manually ("#9701 IP address automatic setting" is set to 0), the same setting will be made for all parameters.

9711 to 9781 parameters:

Set the server information required for using the Ethernet function.

Server information for up to four units can be set.

【#1926(PR)】	Global IP address	IP address
	Set the main CPU's IP address. Set the NC IP address seen from an external source.	
【#1927(PR)】	Global Subnet mask	Subnet mask
	Set the subnet mask for the IP address.	
【#1928(PR)】	Global Gateway	Gateway
	Set the IP address for the gateway.	
【#1934(PR)】	Local IP address	
	Set the HMI side CPU's IP address.	
【#1935(PR)】	Local Subnet mask	
	Set the HMI side CPU's subnet mask.	
【#11005(PR)】	PC IP address	IP address setting
	Set the IP address for the display unit or PC in which machining programs are stored. Set the IP address for the display unit on which the automatic power OFF will be executed. (Note) When "0.0.0.0" is input, "192.168.100.2" is automatically set.	
	*This parameter is used only for 700 Series.	
	PC subnet	
	Set the subnet mask for the display unit or PC in which machining programs are stored.	
	PC Gateway	
	Set the gateway for the display unit or PC in which machining programs are stored.	
【#9701(PR)】	IP addr auto set	
	The IP address is automatically assigned from the server. 0: Manual setting 1: Automatic setting	
	(Note) When the automatic setting is selected, "#11005 PC IP address, PC Subnet, PC Gateway" will be invalid.	

【#9706】 Host No.

Select the No. of the host to be used from host 1 to host 4.

---Setting range---

1 to 4 : Host No.

【#9711】 Host1 host name

Set the host computer name.

This parameter allows the NC to easily recognize the host computer on the network. Set the host computer's name (name registered in C:¥windows¥hosts) or the IP address.

---Setting example---

For host name: mspc160

For IP address: 150.40.0.111

(Note) Set the host computer's TCP/IP address if communication is not carried out correctly.

---Setting range---

15 characters (alphanumeric) or less

【#9712】 Host1 user name

Set the user name when logging into the host computer.

---Setting range---

15 characters (alphanumeric) or less

【#9713】 Host1 password

Set the password when logging into the host computer.

---Setting range---

15 characters (alphanumeric) or less

【#9714】 Host1 directory

Set the directory name of the host computer.

The directory released to the client (NC unit) with the host computer's server is handled as root directory by the NC unit.

---Setting range---

31 characters (alphanumeric) or less

【#9715】 Host1 host type

Select the type of the host computer.

0: UNIX/PC automatic judgment

1: UNIX

2: PC (DOS)

(Note) When "0" is set, the settings for the following parameters will be invalid.

- #9716 Wrd pos: name
- #9717 Wrd pos: size
- #9718 Wrd pos: Dir
- #9719 Wrd pos: cmnt
- #9720 Wrd num: cmnt

【#9716】 Host 1 Wrd pos: name

Set the file name display position (nth word from left) of the list displayed when the ftp command "dir" is executed.

(Note) One word designates a character string divided by one or more spaces.

---Setting range---

0 to 100

0: Default value

【#9717】 Host 1 Wrd pos: size

Set the size display position (nth word from left) of the list displayed when the ftp command "dir" is executed.

(Note) One word designates a character string divided by one or more spaces.

---Setting range---

0 to 100

0: Default value

【#9718】 Host 1 Wrd pos: Dir

Set the <DIR> display position (nth word from left) of the list displayed when the ftp command "dir" is executed.

(Note) One word designates a character string divided by one or more spaces.

---Setting range---

0 to 100

0: Default value

【#9719】 Host 1 Wrd pos: cmnt

Set the comment (date, time, etc.) display position (nth word from left) of the list displayed when the ftp command "dir" is executed.

(Note) One word designates a character string divided by one or more spaces.

---Setting range---

0 to 100

0: Default value

【#9720】 Host 1 Wrd num: cmnt

Set the number of words to be displayed as a comment.

(Note) One word designates a character string divided by one or more spaces.

---Setting range---

0 to 100

0: Default value

【#9721】 Host 1 no total siz

Set whether to display the total number of characters registered in the machining programs of host1 when displaying the file list.

If there are many files in the directory to be referred to, the list can be updated quickly by setting "1".

0: Display

1: Not display

【#9731】 Host2 host name

Set the host computer name.

This parameter allows the NC to easily recognize the host computer on the network. Set the host computer's name (name registered in C:%windows%hosts) or the IP address.

---Setting example---

For host name: mspc160

For IP address: 150.40.0.111

(Note) Set the host computer's TCP/IP address if communication is not carried out correctly.

---Setting range---

15 characters (alphanumeric) or less

【#9732】 Host2 user name

Set the user name when logging into the host computer.

---Setting range---

15 characters (alphanumeric) or less

【#9733】 Host2 password

Set the password when logging into the host computer.

---Setting range---

15 characters (alphanumeric) or less

[#9734] Host2 directory

Set the directory name of the host computer.
The directory released to the client (NC unit) with the host computer's server is handled as the root directory by the NC unit.

---Setting range---

31 characters (alphanumeric) or less

 [#9735] Host2 host type

Select the type of the host computer.

0: UNIX/PC automatic judgment

1: UNIX

2: PC (DOS)

(Note) When "0" is set, the settings for the following parameters will be invalid.

- #9736 Wrd pos: name
- #9737 Wrd pos: size
- #9738 Wrd pos: Dir
- #9739 Wrd pos: cmnt
- #9740 Wrd num: cmnt

 [#9736] Host 2 Wrd pos: name

Set the file name display position (nth word from left) of the list displayed when the ftp command "dir" is executed.

(Note) One word designates a character string divided by one or more spaces.

---Setting range---

0 to 100

0: Default value

 [#9737] Host 2 Wrd pos: size

Set the size display position (nth word from left) of the list displayed when the ftp command "dir" is executed.

(Note) One word designates a character string divided by one or more spaces.

---Setting range---

0 to 100

0: Default value

 [#9738] Host 2 Wrd pos: Dir

Set the <DIR> display position (nth word from left) of the list displayed when the ftp command "dir" is executed.

(Note) One word designates a character string divided by one or more spaces.

---Setting range---

0 to 100

0: Default value

 [#9739] Host 2 Wrd pos: cmnt

Set the comment (date, time, etc.) display position (nth word from left) of the list displayed when the ftp command "dir" is executed.

(Note) One word designates a character string divided by one or more spaces.

---Setting range---

0 to 100

0: Default value

 [#9740] Host 2 Wrd num: cmnt

Set the number of words to be displayed as a comment.

(Note) One word designates a character string divided by one or more spaces.

---Setting range---

0 to 100

0: Default value

【#9741】 Host 2 no total siz

Set whether to display the total number of characters registered in the machining programs of host1 when displaying the file list.
If there are many files in the directory to be referred to, the list can be updated quickly by setting "1".

---Setting range---

0: Display
1: Not display

【#9751】 Host3 host name

Set the host computer name.
This parameter allows the NC to easily recognize the host computer on the network. Set the host computer's name (name registered in C:¥windows¥hosts) or the IP address.

---Setting example---

For host name: mspc160
For IP address: 150.40.0.111

(Note) Set the host computer's TCP/IP address if communication is not carried out correctly.

---Setting range---

15 characters (alphanumeric) or less

【#9752】 Host3 user name

Set the user name when logging into the host computer.

---Setting range---

15 characters (alphanumeric) or less

【#9753】 Host3 password

Set the password when logging into the host computer.

---Setting range---

15 characters (alphanumeric) or less

【#9754】 Host3 directory

Set the directory name of the host computer.
The directory released to the client (NC unit) with the host computer's server is handled as the root directory by the NC unit.

---Setting range---

31 characters (alphanumeric) or less

【#9755】 Host3 host type

Select the type of the host computer.

0: UNIX/PC automatic judgment
1: UNIX
2: PC (DOS)

(Note) When "0" is set, the settings for the following parameters will be invalid.

- #9756 Wrd pos: name
- #9757 Wrd pos: size
- #9758 Wrd pos: Dir
- #9759 Wrd pos: cmnt
- #9760 Wrd num: cmnt

【#9756】 Host 3 Wrd pos: name

Set the file name display position (nth word from left) of the list displayed when the ftp command "dir" is executed.

(Note) One word designates a character string divided by one or more spaces.

---Setting range---

0 to 100
0: Default value

【#9757】 Host 3 Wrd pos: size

Set the size display position (nth word from left) of the list displayed when the ftp command "dir" is executed.

(Note) One word designates a character string divided by one or more spaces.

---Setting range---

0 to 100

0: Default value

【#9758】 Host 3 Wrd pos: Dir

Set the <DIR> display position (nth word from left) of the list displayed when the ftp command "dir" is executed.

(Note) One word designates a character string divided by one or more spaces.

---Setting range---

0 to 100

0: Default value

【#9759】 Host 3 Wrd pos: cmnt

Set the comment (date, time, etc.) display position (nth word from left) of the list displayed when the ftp command "dir" is executed.

(Note) One word designates a character string divided by one or more spaces.

---Setting range---

0 to 100

0: Default value

【#9760】 Host 3 Wrd num: cmnt

Set the number of words to be displayed as a comment.

(Note) One word designates a character string divided by one or more spaces.

---Setting range---

0 to 100

0: Default value

【#9761】 Host 3 no total siz

Set whether to display the total number of characters registered in the machining programs of host1 when displaying the file list.

If there are many files in the directory to be referred to, the list can be updated quickly by setting "1".

---Setting range---

0: Display

1: Not display

【#9771】 Host4 host name

Set the host computer name.

This parameter allows the NC to easily recognize the host computer on the network. Set the host computer's name (name registered in C:¥windows¥hosts) or the IP address.

---Setting example---

For host name: mspc160

For IP address: 150.40.0.111

(Note) Set the host computer's TCP/IP address if communication is not carried out correctly.

---Setting range---

15 characters (alphanumeric) or less

【#9772】 Host4 user name

Set the user name when logging into the host computer.

---Setting range---

15 characters (alphanumeric) or less

[#9773] Host4 password

Set the password when logging into the host computer.

---Setting range---

15 characters (alphanumeric) or less

 [#9774] Host4 directory

Set the directory name of the host computer.

The directory released to the client (NC unit) with the host computer's server is handled as the root directory by the NC unit.

---Setting range---

31 characters (alphanumeric) or less

 [#9775] Host4 host type

Select the type of the host computer.

0: UNIX/PC automatic judgment

1: UNIX

2: PC (DOS)

(Note) When "0" is set, the settings for the following parameters will be invalid.

- #9776 Wrd pos: name
- #9777 Wrd pos: size
- #9778 Wrd pos: Dir
- #9779 Wrd pos: cmnt
- #9780 Wrd num: cmnt

 [#9776] Host 4 Wrd pos: name

Set the file name display position (nth word from left) of the list displayed when the ftp command "dir" is executed.

(Note) One word designates a character string divided by one or more spaces.

---Setting range---

0 to 100

0: Default value

 [#9777] Host 4 Wrd pos: size

Set the size display position (nth word from left) of the list displayed when the ftp command "dir" is executed.

(Note) One word designates a character string divided by one or more spaces.

---Setting range---

0 to 100

0: Default value

 [#9778] Host 4 Wrd pos: Dir

Set the <DIR> display position (nth word from left) of the list displayed when the ftp command "dir" is executed.

(Note) One word designates a character string divided by one or more spaces.

---Setting range---

0 to 100

0: Default value

 [#9779] Host 4 Wrd pos: cmnt

Set the comment (date, time, etc.) display position (nth word from left) of the list displayed when the ftp command "dir" is executed.

(Note) One word designates a character string divided by one or more spaces.

---Setting range---

0 to 100

0: Default value

【#9780】 Host 4 Wrđ num: cmnt

Set the number of words to be displayed as a comment.
(Note) One word designates a character string divided by one or more spaces.

---Setting range---

0 to 100
0: Default value

【#9781】 Host 4 no total siz

Set whether to display the total number of characters registered in the machining programs of host1 when displaying the file list.
If there are many files in the directory to be referred to, the list can be updated quickly by setting "1".

---Setting range---

0: Display
1: Not display

- (Note 1)** The user name and password are required when logging in.
- (Note 2)** It is necessary to enable reading/writing when exchanging files.
- (Note 3)** With the Personal WEB Server and Windows NT 4.0 fpt Server, the file list format can be selected from DOS or UNIX.
- (Note 4)** The directory released to the client (NC unit) with the host computer's server is handled as the root directory by the NC unit.

9.7 Computer Link Parameters

【#9601】 BAUD RATE

Select the rate at which data is transferred.

---Setting range---

- 0: 19200 (bps)
- 1: 9600
- 2: 4800
- 3: 2400
- 4: 1200
- 5: 600
- 6: 300
- 7: 110
- 8: 38400

【#9602】 STOP BIT

Select the stop bit length used in the start-stop system.

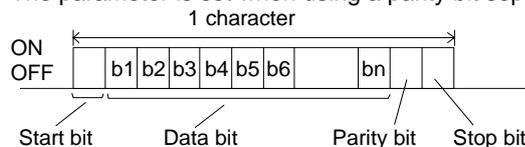
Refer to "#9603 PARITY EFFECTIVE". At the output of data, the number of characters is always adjusted to for the parity check.

- 1: 1 (bit)
- 2: 1.5
- 3: 2

【#9603】 PARITY EFFECTIVE

Select whether to add the parity bit to the data.

The parameter is set when using a parity bit separately from the data bit.



Set this parameter according to the specifications of input/output device.

- 0: Not add a parity bit at the input/output
- 1: Add a parity bit at the input/output

【#9604】 EVEN PARITY

Select odd or even when parity is added to the data. This parameter is ignored when no parity is added.

- 0: Odd parity
- 1: Even parity

【#9605】 CHR. LENGTH

Select the length of the data bit.

Refer to "#9603 PARITY EFFECTIVE".

- 0: 5 (bit)
- 1: 6
- 2: 7 (NC connection not supported)
- 3: 8

【#9606】 HAND SHAKE

Select the transmission control method.

"3" (DC code method) should be set for computer link B.

- 0: No control
- 1: RTS/CTS method
- 2: No handshaking
- 3: DC code method

【#9607】 TIME-OUT SET

Set the time-out time at which an interruption of data transfer during data input/output should be detected.

"0" means infinite time-out.

---Setting range---

0 to 999 (1/10s)

【#9608】 DATA CODE

Set the code to be used for the data description.

Refer to "#9603 PARITY EFFECTIVE".

0: ASCII code

1: ISO code

【#9609】 LINK PARAM. 1**bit1: DC1 output after NAK or SYN**

Select whether to output the DC1 code after the NAK or SYN code is output.

0: Not output the DC1 code.

1: Output the DC1 code.

bit7: Enable/disable resetting

Select whether to enable the resetting in the computer link.

0: Enable

1: Disable

【#9610】 LINK PARAM. 2**Bit 2: Specify the control code parity (even parity for the control code).**

Select whether to add an even parity to the control code, in accordance with the I/O device specifications.

0: Not add a parity bit to the control code

1: Add a parity bit to the control code

Bit 3: Parity V

Select whether to enable checking of parity V in one block at the input of the data.

0: Disable

1: Enable

【#9611】 Link PARAM. 3

Not used. Set to "0".

【#9612】 Link PARAM. 4

Not used. Set to "0".

【#9613】 Link PARAM. 5

Not used. Set to "0".

【#9614】 START CODE

Select the code used to command the first transfer of file data.

This parameter is used for a specific user. Normally set "0".

0: DC1 (11H)

1: BEL (07H)

【#9615】	CTRL. CODE OUT
	bit 0: NAK output Select whether to send the NAK code to the host if a communication error occurs in computer link B. 0: Not output the NAK code 1: Output the NAK code.
	bit 1: SYN output Select whether to send the SYN code to the host if NC resetting or an emergency stop occurs in computer link B. 0: Not output the SYN code. 1: Output the SYN code.
	bit 3: DC3 output Select whether to send the DC3 code to the host when the communication ends in computer link B. 0: Not output the DC3 code. 1: Output the DC3 code.
【#9616】	CTRL. INTERVAL Not used. Set to "0".
【#9617】	WAIT TIME Not used. Set to "0".
【#9618】	PACKET LENGTH Not used. Set to "0".
【#9619】	BUFFER SIZE Not used. Set to "0".
【#9620】	START SIZE Not used. Set to "0".
【#9621】	DC1 OUT SIZE Not used. Set to "0".
【#9622】	POLLING TIMER Not used. Set to "0".
【#9623】	TRANS. WAIT TMR Not used. Set to "0".
【#9624】	RETRY COUNTER Not used. Set to "0".

9.8 Subprogram Storage Destination Parameters

【#8880】 Subpro stor D0: dev

Select the storage destination (device) for the subprogram.
When D0 is designated at a subprogram call, the subprogram to be called will be searched from the device selected with this parameter.

(Example)

The following will be searched:

M98 P (program No.), D0

Device: "#8880 Subpro stor D0: dev" device

Directory: "#8881 Subpro stor D0: dir" directory

(Note 1) When the called subprogram is not found in the selected storage destination, a program error will occur.

(Note 2) When D0 to D4 is not designated at a subprogram call, the subprogram will be searched from the memory.

---Setting range---

Device name

Setting value	Display name
M	Memory
G	HD
F	FD
R	Memory card
D	Data server
E	Ethernet

【#8881】 Subpro stor D0: dir

Select the storage destination (directory) for the subprogram.

When D0 is designated at a subprogram call, the subprogram to be called will be searched from the directory selected with this parameter.

Refer to "#8880 Subpro stor D0: dev".

---Setting range---

Directory 48 characters

【#8882】 Subpro stor D1: dev

Select the storage destination (device) for the subprogram.

When D1 is designated at a subprogram call, the subprogram to be called will be searched from the device selected with this parameter.

(Example)

The following will be searched:

M98 P (program No.), D1

Device: "#8882 Subpro stor D1: dev" device

Directory: "#8883 Subpro stor D1: dir" directory

(Note 1) When the called subprogram is not found in the selected storage destination, a program error will occur.

(Note 2) When D0 to D4 is not designated at a subprogram call, the subprogram will be searched from the memory.

---Setting range---

Setting value	Display name
M	Memory
G	HD
F	FD
R	Memory card
D	Data server
E	Ethernet

【#8883】 Subpro stor D1: dir

Select the storage destination (directory) for the subprogram.

When D1 is designated at a subprogram call, the subprogram to be called will be searched from the directory selected with this parameter.

Refer to "#8882 Subpro stor D1: dev".

---Setting range---

Directory 48 characters

【#8884】 Subpro stor D2: dev

Select the storage destination (device) for the subprogram.

When D2 is designated at a subprogram call, the subprogram to be called will be searched from the device selected with this parameter.

(Example)

The following will be searched:

M98 P (program No.), D2

Device: "#8884 Subpro stor D2: dev" device

Directory: "#8885 Subpro stor D2: dir" directory

(Note 1) When the called subprogram is not found in the selected storage destination, a program error will occur.

(Note 2) When D0 to D4 is not designated at a subprogram call, the subprogram will be searched from the memory.

---Setting range---

Setting value	Display name
M	Memory
G	HD
F	FD
R	Memory card
D	Data server
E	Ethernet

【#8885】 Subpro stor D2: dir

Select the storage destination (directory) for the subprogram.

When D2 is designated at a subprogram call, the subprogram to be called will be searched from the directory selected with this parameter.

Refer to "#8884 Subpro stor D2: dev".

---Setting range---

Directory 48 characters

【#8886】 Subpro stor D3: dev

Select the storage destination (device) for the subprogram.

When D3 is designated at a subprogram call, the subprogram to be called will be searched from the device selected with this parameter.

(Example)

The following will be searched:

M98 P (program No.), D3

Device: "#8886 Subpro stor D3: dev" device

Directory: "#8887 Subpro stor D3: dir" directory

(Note 1) When the called subprogram is not found in the selected storage destination, a program error will occur.

(Note 2) When D0 to D4 is not designated at a subprogram call, the subprogram will be searched from the memory.

---Setting range---

Setting value	Display name
M	Memory
G	HD
F	FD
R	Memory card
D	Data server
E	Ethernet

【#8887】 Subpro stor D3: dir

Select the storage destination (directory) for the subprogram.

When D3 is designated at a subprogram call, the subprogram to be called will be searched from the directory selected with this parameter.

Refer to "#8886 Subpro stor D3: dev".

---Setting range---

Directory 48 characters

【#8888】 Subpro stor D4: dev

Select the storage destination (device) for the subprogram.

When D4 is designated at a subprogram call, the subprogram to be called will be searched from the device selected with this parameter.

(Example)

The following will be searched:

M98 P (program No.), D4

Device: "#8888 Subpro stor D4: dev" device

Directory: "#8889 Subpro stor D4: dir" directory

(Note 1) When the called subprogram is not found in the selected storage destination, a program error will occur.

(Note 2) When D0 to D4 is not designated at a subprogram call, the subprogram will be searched from the memory.

---Setting range---

Device name

Setting value	Display name
M	Memory
G	HD
F	FD
R	Memory card
D	Data server
E	Ethernet

【#8889】 Subpro stor D4: dir

Select the storage destination (directory) for the subprogram.

When D4 is designated at a subprogram calling, the subprogram to be called will be searched from the directory selected with this parameter.

Refer to "#8888 Subpro stor D4: dev".

---Setting range---

Directory 48 characters

9.9 Axis Parameters

Set up the parameter required for each axis.

【#1063】	mandog Manual dog-type	Select the manual reference position return method for the second return (after the coordinate system is established) and later. The initial reference position return after the power ON is performed with dog-type return, and the coordinate system will be established. (This setting is not required when the absolute position detection is used.) 0: High speed return 1: Dog-type
【#8201】	AX. RELEASE	Select the function to remove the control axis from the control target. 0: Control as normal. 1: Remove from control target.
【#8202】	OT-CHECK OFF	Select whether to enable the stored stroke limit II function set in #8204 and #8205. 0: Enable 1: Disable
【#8203】	OT-CHECK-CANCEL	When the simple absolute position method ("#2049 type" is "9") is selected, the stored stroke limits I, II (or IIB) and IB can be disabled until the first reference position return is executed after the power is turned ON. 0: Enable (according to #8202) 1: Temporarily cancel (Note) "#8203 OT-CHECK-CANCEL" affects all the stored stroke limits.
【#8204】	OT-CHECK-N	Set the coordinates of the (-) direction in the movable range of the stored stroke limit II or the lower limit coordinates of the prohibited range of stored stroke limit IIB. If the sign and value are the same as #8205, the stored stroke limit II (or IIB) will be invalid. If the stored stroke limit IIB function is selected, the prohibited range will be between two points even when #8204 and #8205 are set in reverse. When II is selected, the entire range will be prohibited if #8204 and #8205 are set in reverse. ---Setting range--- ±99999.999 (mm)
【#8205】	OT-CHECK-P	Set the coordinates of the (+) direction in the movable range of the stored stroke limit II or the upper limit coordinates of the prohibited range of stored stroke limit IIB. ---Setting range--- ±99999.999 (mm)
【#8206】	TOOL CHG. P	Set the coordinates of the tool change position for G30. n (tool change position return). Set with coordinates in the basic machine coordinate system. ---Setting range--- ±99999.999 (mm)
【#8207】	G76/87 IGNR for M system only	Select whether to enable the shift operation at G76 (fine boring) and G87 (back boring). 0: Enable 1: Disable

【#8208】 G76/87 (-) for M system only

Select the shift direction at G76 and G87.
0: Shift to (+) direction
1: Shift to (-) direction

【#8209】 G60 SHIFT for M system only

Set the last positioning direction and distance for a G60 (unidirectional positioning) command.

---Setting range---
±99999.999 (mm)

【#8210】 OT INSIDE

Select whether the stored stroke limit function set by #8204 and #8205 prevents the machine from moving to the inside or outside of the specified range.
0: Inhibits outside area (Select stored stroke limit II.)
1: Inhibits inside area (Select stored stroke limit II B.)

【#8211】 MIRR. IMAGE

Select whether to enable the parameter mirror image function.
0: Disable
1: Enable

【#8213(PR)】 Rotation axis type

Select the rotation type (short-cut valid/invalid) or linear type (workpiece coordinate linear type/all coordinate linear type).

This parameter is enabled only when "#1017 rot" is set to "1". (Note)

0: Short-cut invalid

1: Short-cut valid

2: Workpiece coordinate linear type

3: All coordinate linear type

(Note) The movement method is as follows by the specified rotation axis type.

Setting value	0	1	2	3
Workpiece coordinate value	Display range: 0° to 359.999°		Display range: 0° to ±99999.999°	
Machine coordinate value/relative position	Display range: 0° to 359.999°		Display range: 0° to ±99999.999°	
ABS command	The incremental amount from the end point to the current position is divided by 360, and the axis moves by the remainder amount according to the sign.	Moves with a short-cut to the end point.	In the same manner as the normal linear axis, moves according to the sign by the amount obtained by subtracting the current position from the end point.	
INC command	Moves in the direction of the commanded sign by the commanded incremental amount starting at the current position.			
Reference position return	The movement to the middle point applies to the ABS command or the INC command.			
	Returns with movement within 360 degrees from the middle point to reference position.		Moves and returns in the reference position direction for the difference from the current position to the reference position.	

【#8215】 TLM std length

Set the TLM standard length.

TLM standard length is the distance from a tool replacement point (reference position) to the measurement basic point (surface) which is used to measure the tool length.

---Setting range---

-99999.999 to 99999.999 (mm)

【#8216】 Type in G28 return

Select the performance after establishing the reference position in reference position return command.

0: Moves to the reference position.

1: Won't move to the reference position.

【#8217】 Check start point

Set a drawing start position in graphic check of each axis.

---Setting range---

-99999.999 to 99999.999 (mm)

9.10 Barrier Data (For L System Only)

【#8300】 P0

Set the reference X-coordinates of the chuck and the tail stock barrier.
Set the center coordinate (radius value) of workpiece by the basic machine coordinate system.

---Setting range---
±99999.999 (mm)

【#8301】 P1

Set the area of the chuck and tail stock barrier.
Set the coordinate from the center of workpiece (P0) for X-axis. (radius value)
Set the coordinate value by basic machine coordinate system for Z-axis.

---Setting range---
±99999.999 (mm)

【#8302】 P2

Set the area of the chuck and tail stock barrier.
Set the coordinate from the center of workpiece (P0) for X-axis. (radius value)
Set the coordinate value by basic machine coordinate system for Z-axis.

---Setting range---
±99999.999 (mm)

【#8303】 P3

Set the area of the chuck and tail stock barrier.
Set the coordinate from the center of workpiece (P0) for X-axis. (radius value)
Set the coordinate value by basic machine coordinate system for Z-axis.

---Setting range---
±99999.999 (mm)

【#8304】 P4

Set the area of the chuck and tail stock barrier.
Set the coordinate from the center of workpiece (P0) for X-axis. (radius value)
Set the coordinate value by basic machine coordinate system for Z-axis.

---Setting range---
±99999.999 (mm)

【#8305】 P5

Set the area of the chuck and tail stock barrier.
Set the coordinate from the center of workpiece (P0) for X-axis. (radius value)
Set the coordinate value by basic machine coordinate system for Z-axis.

---Setting range---
±99999.999 (mm)

【#8306】 P6

Set the area of the chuck and tail stock barrier.
Set the coordinate from the center of workpiece (P0) for X-axis. (radius value)
Set the coordinate value by basic machine coordinate system for Z-axis.

---Setting range---
±99999.999 (mm)

【#8311】 P7

Set the area of the left spindle section.
Set the coordinate from the center of workpiece (P0) for X-axis. (radius value)
Set the coordinate value by basic machine coordinate system for Z-axis.

---Setting range---
±99999.999 (mm)

【#8312】 P8

Set the area of the left spindle section.
 Set the coordinate from the center of workpiece (P0) for X-axis. (radius value)
 Set the coordinate value by basic machine coordinate system for Z-axis.

---Setting range---
 ±99999.999 (mm)

【#8313】 P9

Set the area of the right spindle section.
 X axis: Set the coordinate from the workpiece center (P0). (radius value)
 Z axis: Set the coordinates in the basic machine coordinate system.

---Setting range---
 ±99999.999 (mm)

【#8314】 P10

Set the area of the right spindle section.
 Set the coordinate from the center of workpiece (P0) for X-axis. (radius value)
 Set the coordinate value by basic machine coordinate system for Z-axis.

---Setting range---
 ±99999.999 (mm)

【#8310】 Barrier ON

Select whether to enable the chuck and tailstock barrier.
 0: Disable (Setting from special display unit will be enabled)
 1: Enable

【#8315】 Barrier Type (L)

Select the shape of the left chuck and tailstock barrier.
 0: No area
 1: Chuck
 2: Tailstock

【#8316】 Barrier Type (R)

Select the shape of the right chuck and tailstock barrier.
 0: No area
 1: Chuck
 2: Tailstock

【#8317】 ELIV. AX. Name

Set the name of the delivery axis when the right chuck and tailstock barrier is movable.
 When using the multi-part system method and the delivery axis is an axis in the other part system, designate the axis including the part system as 1A, 1B or 2A, 2B. If the part system is not designated as A and B, the set part system will be used.

---Setting range---
 A/B/.. (axis name)
 1A/1B/..
 2A/2B/.. (with part system designated)
 0: Cancel

【#8318】 Stock Angle (L)

Set the angle for the left tailstock end section.
 The angle will be interpreted as 90° if there is no setting (when "0" is set).

---Setting range---
 0 to 180 (°)
 0: 90° (default)

【#8319】 Stock Angle (R)

Set the angle for the right tailstock end section.

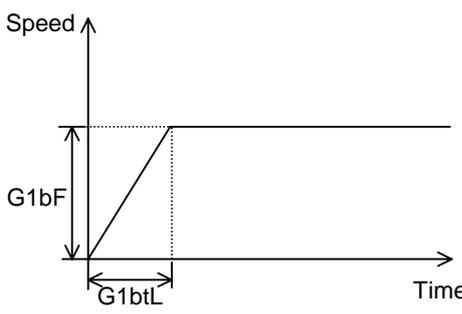
The angle will be interpreted as 90° if there is no setting (when "0" is set).

---Setting range---

0 to 180 (°)

0: 90° (default)

9.11 High Accuracy Parameters

- 【#1149】 cireft Arc deceleration speed change**
 Select whether to decelerate at the arc entrance or exit.
 0: Not decelerate
 1: Decelerate
- 【#1205】 G0bdcc Acceleration and deceleration before G0 interpolation**
 0: Post-interpolation acceleration/deceleration is applied to G00.
 1: Pre-interpolation acceleration/deceleration is applied to G00 even in the high accuracy control mode.
 2: Rapid traverse constant inclination multi-step acceleration/deceleration is enabled.
 (Note) "1" cannot be set for the 2nd part system and the following.
- 【#1206】 G1bF Maximum speed**
 Set a cutting feedrate when applying pre-interpolation acceleration/deceleration.
 When high-accuracy control time constant expansion is valid, set the maximum of cutting feed clamp speed of each axis.
 ---Setting range---
 1 to 999999 (mm/min)
- 【#1207】 G1btL Time constant**
 Set a cutting feed time constant when applying pre-interpolation acceleration/deceleration.
 When "0" is set, the time constant will be clamped at 1ms.
- 
- Setting range---
 Without high-accuracy control time constant expansion : 0 to 5000 (ms)
 With high-accuracy control time constant expansion : 0 to 30000 (ms)
- 【#1209】 cirdcc Arc deceleration speed**
 Set the deceleration speed at the arc entrance or exit.
 ---Setting range---
 1 to 999999 (mm/min)
- 【#1568】 SfiltG1 G01 soft acceleration/deceleration filter**
 Set the filter time constant for smoothly changing the acceleration rate for the cutting feed acceleration/deceleration in pre-interpolation acceleration/deceleration.
 ---Setting range---
 0 to 200 (ms)
- 【#1569】 SfiltG0 G00 soft acceleration/deceleration filter**
 Set the filter time constant for smoothly changing the acceleration rate for the rapid traverse acceleration/deceleration in pre-interpolation acceleration/deceleration.
 ---Setting range---
 0 to 200 (ms)

【#1570】 Sfilt2 Soft acceleration/deceleration filter 2

Set the filter time constant for smoothly changing the acceleration rate in pre-interpolation acceleration/deceleration.
This will be invalid when "0" or "1" is set.

---Setting range---
0 to 26 (ms)

【#1571】 SSSdis SSS control adjustment coefficient fixed value selection

Fix the shape recognition range for SSS control.

<PRECISION> (High-accuracy control)**【#8019】 R COMP**

Set a compensation coefficient for reducing a control error in the reduction of a corner roundness and arch radius.

The larger the setup value, the smaller the theoretical error will be. However, since the speed at the corner will go down, the cycle time will be extended.

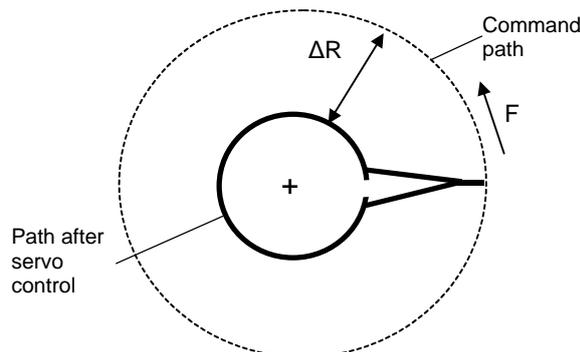
Coefficient = 100 - setting value

(Note) This is valid when "#8021 COMP CHANGE" is set to "0".

---Setting range---
0 to 99 (%)

Theor R decrease

The value calculated with the following data is displayed for the theoretical radius reduction error amount ΔR (mm).

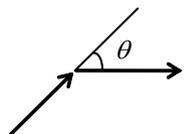


Theoretical radius reduction amount at arc center

【#8020】 DCC. angle

Set up the minimum value of an angle (external angle) that should be assumed to be a corner.

When an inter-block angle (external angle) in high-accuracy mode is larger than the set value, it will be determined as a corner and the speed will go down to sharpen the edge.



If the set value is smaller than θ , the speed goes down to optimize the corner.

(Note) If "0" is set, it will be handled as 5 degrees.

---Setting range---
0 to 89 (degrees)
0: The angle will be 5°.

【#8021】 COMP_CHANGE

Select whether to share or separate the compensation coefficient at the corner/curve during the high-accuracy control mode.

0: Share ("#8019 R COMP" is applied.)

1: Separate

- Corner : #8022 CORNER COMP

- Curve : #8023 CURVE COMP

(Note) Set "1" when using SSS control.

【#8022】 CORNER COMP

Set the compensation coefficient to further reduce or increase the roundness at the corner during the high-accuracy control mode.

Coefficient = 100 - setting value

(Note) This is valid when "#8021 COMP CHANGE" is set to "1".

---Setting range---

-1000 to 99 (%)

【#8023】 CURVE COMP

Set the compensation coefficient to further reduce or increase the radius reduction amount at the curve (arc, involute, spline) during the high-accuracy control mode.

Coefficient = 100 - setting value

(Note) This is valid when "#8021 COMP CHANGE" is set to "1".

For theoretical radius reduction error amount, refer to "Theor R decrease" in "#8019 R COMP"

---Setting range---

-1000 to 99 (%)

<SPILINE> (High-accuracy spline)**【#8025】 SPLINE ON for M system only**

Select whether to enable the spline function.

0: Disable

1: Enable

Spline interpolation is valid during G61.2 modal, regardless of this setting.

【#8026】 CANCEL ANG. for M system only

Set the angle where the spline interpolation is temporarily canceled.

When the angle made by blocks exceeds this parameter setting value, spline interpolation will be canceled temporarily. In consideration of the pick feed, set a value a little smaller than the pick feed angle.

---Setting range---

0 to 180 (°)

0: 180 (°)

【#8027】 Toler-1 for M system only

Set the maximum chord error (tolerance) in a block that includes an inflection point. Set the tolerance applicable when the applicable block is developed to fine segments by CAM. (normally about 10 μm)
When "0.000" is set, the applicable block will be linear.

---Setting range---

0.000 to 100.000 (mm)

【#8028】 Toler-2 for M system only

Set the maximum chord error (tolerance) in a block that includes no inflection point. Set the tolerance applicable when the applicable block is developed to fine segments by CAM. (normally about 10 μm)
When "0.000" is set, the applicable block will be linear.

---Setting range---

0.000 to 100.000 (mm)

【#8029】 FairingL for M system only

Set the length of the block subject to fairing.
(Enabled when "#8033 Fairing ON" is set to "1".)

---Setting range---
0 to 100.000 (mm)

【#8030】 MINUTE LENG for M system only

Set the fine-segment length where the spline interpolation is temporarily canceled.
When the length of one block exceeds this parameter setting value, spline interpolation is canceled temporarily and linear interpolation is performed. Set a value a little smaller than one block length of the program.
If "-1" is set, spline interpolation will be performed regardless of block length.

---Setting range---
-1 to 127 (mm)
0: 1 (mm)

<Fairing>

【#8033】 Fairing ON for M system only

Select whether to use the fairing function.
0: Not use
1: Use
Fairing function is enabled during G61.2 modal, regardless of this setting.

【#8034】 AccClamp ON for M system only

Select the method for clamping the cutting speed.
0: Clamp with parameter "#2002 clamp" or the corner deceleration function.
1: Clamp the cutting speed with acceleration judgment.
(Enabled when "#8033 Fairing ON" is set to "1".)

【#8036】 CordecJudge for M system only

Select the condition to decide a corner.
0: A corner is decided from the angle of the neighboring block.
1: A corner is decided from the angle of the neighboring block, excluding minute blocks.
(Enabled when "#8033 Fairing ON" is set to "1".)

【#8037】 CorJudgeL for M system only

Set the length of the block to be excluded when deciding a corner.
(Enabled when "#8036 CordecJudge" is set to "1".)

---Setting range---
0 to 99999.999 (mm)

<SSS control>

【#8090】 SSS ON for M system only

Set whether to enable the SSS control with G05 P10000.
0: Disable
1: Enable

【#8091】 StdLength for M system only

Set the maximum value of the range for recognizing the shape.
To eliminate the effect of steps or errors, etc., set a large value. To enable sufficient deceleration, set a small value.
If "0.000" is set, the standard value (1.000mm) will be applied.

---Setting range---
0 to 100.000 (mm)

【#8092】 ClampCoeff for M system only

Set the clamp speed at the curved section configured of fine segments.
Coefficient = $\sqrt{\text{setting value}}$

---Setting range---
1 to 100

【#8093】 StepLeng for M system only

Set the width of the step at which the speed is not to be decelerated. (Approximately the same as the CAM path difference [Tolerance].)
If "0" is set, the standard value (5 μ m) will be applied.
If a minus value is set, the speed will decelerate at all minute steps.

---Setting range---
-1.000 to 0.100 (mm)

【#8094】 DccWaitAdd for M system only

Set the time to wait for deceleration when the speed FB does not drop to the clamp speed.

---Setting range---
0 to 100 (ms)

9.12 High-accuracy Axis Parameters

 [#2001]	rapid	Rapid traverse rate	<p>Set the rapid traverse feedrate for each axis. (Note) The maximum value to be set depends on the machine specifications.</p> <p>---Setting range--- 1 to 1000000 (mm/min)</p>
 [#2002]	clamp	Cutting feedrate for clamp function	<p>Set the maximum cutting feedrate for each axis. Even if the feedrate in G01 exceeds this value, the clamp will be applied at this feedrate.</p> <p>---Setting range--- 1 to 1000000 (mm/min)</p>
 [#2010]	fwd_g	Feed forward gain	<p>Set a feed forward gain for pre-interpolation acceleration/deceleration. The larger the set value, the smaller the theoretical control error will be. However, if a machine vibration occurs, set the smaller value.</p> <p>---Setting range--- 0 to 200 (%)</p>
 [#2068]	G0fwdg	G00 feed forward gain	<p>Set a feed forward gain for G00 pre-interpolation acceleration/deceleration. The larger the setting value, the shorter the positioning time during in-position checking. If a machine vibration occurs, set the smaller value.</p> <p>---Setting range--- 0 to 200 (%)</p>
 [#2096]	crncsp	Minimum corner deceleration speed	<p>Set the minimum clamp speed for corner deceleration in the high-accuracy control mode. Normally set "0". (Note) This parameter is invalid during SSS control.</p> <p>---Setting range--- 0 to 1000000 (mm/min)</p>
 [#2109]	Rapid (H-precision)	Rapid traverse rate for high-accuracy control mode	<p>Set the rapid traverse rate for each axis in the high-accuracy control mode. "#2001 rapid" will be used when "0" is set.</p> <p>---Setting range--- 0 to 1000000 (mm/min)</p>
 [#2110]	Clamp (H-precision)	Cutting feed clamp speed for high-accuracy control mode	<p>Set the cutting feed maximum speed for each axis in the high-accuracy control mode. "#2002 clamp" will be used when "0" is set.</p> <p>---Setting range--- 0 to 1000000 (mm/min)</p>

9.13 Operation Parameters

【#8901】 Counter type 1

Set the type of counter displayed at the upper left of the AUTO/MDI display on the Monitor screen.

- 1: Current position
- 2: Work coordinate position
- 3: Machine position
- 4: Program position
- 8: Remain command
- 9: Manual interrupt amount
- 10: Next command
- 11: Restart position
- 12: Remain distance
- 16: Tip work coordinate position
- 18: Tool axis movement
- 19: Tip machine position
- 20: Relative position

---Setting range---

0 to 255

【#8902】 Counter type 2

Set the type of counter displayed at the lower left of the AUTO/MDI display on the Monitor screen.

- 1: Current position
- 2: Work coordinate position
- 3: Machine position
- 4: Program position
- 8: Remain command
- 9: Manual interrupt amount
- 10: Next command
- 11: Restart position
- 12: Remain distance
- 16: Tip work coordinate position
- 18: Tool axis movement
- 19: Tip machine position
- 20: Relative position

---Setting range---

0 to 255

【#8903】 Counter type 3

Set the type of counter displayed at the upper right of the AUTO/MDI display on the Monitor screen.

- 1: Current position
- 2: Work coordinate position
- 3: Machine position
- 4: Program position
- 8: Remain command
- 9: Manual interrupt amount
- 10: Next command
- 11: Restart position
- 12: Remain distance
- 16: Tip work coordinate position
- 18: Tool axis movement
- 19: Tip machine position
- 20: Relative position

---Setting range---

0 to 255

[#8904] Counter type 4

Set the type of counter displayed at the lower right of the AUTO/MDI display on the Monitor screen.

- 1: Current position
- 2: Work coordinate position
- 3: Machine position
- 4: Program position
- 8: Remain command
- 9: Manual interrupt amount
- 10: Next command
- 11: Restart position
- 12: Remain distance
- 16: Tip work coordinate position
- 18: Tool axis movement
- 19: Tip machine position
- 20: Relative position

---Setting range---

0 to 255

 [#8905] Counter type 5

Set the type of counter displayed at the left of the Manual display on the Monitor screen.

- 1: Current position
- 2: Work coordinate position
- 3: Machine position
- 4: Program position
- 8: Remain command
- 9: Manual interrupt amount
- 10: Next command
- 11: Restart position
- 12: Remain distance
- 16: Tip work coordinate position
- 18: Tool axis movement
- 19: Tip machine position
- 20: Relative position

---Setting range---

0 to 255

 [#8906] Counter type 6

Set the type of counter displayed at the right of the Manual display on the Monitor screen.

- 1: Current position
- 2: Work coordinate position
- 3: Machine position
- 4: Program position
- 8: Remain command
- 9: Manual interrupt amount
- 10: Next command
- 11: Restart position
- 12: Remain distance
- 16: Tip work coordinate position
- 18: Tool axis movement
- 19: Tip machine position
- 20: Relative position

---Setting range---

0 to 255

 [#8910] Edit undo

Set whether to enable the Undo function during program edit on the Monitor screen or Edit screen.

- 0: Disable
- 1: Enable

【#8914】 Auto Top search

Select the operation method for restart search type 2.
 0: It is necessary to set the top search position arbitrarily.
 1: The restart search is executed from O No. that is designated as head.

【#8915】 Auto backup day 1

When the NC power is ON after the designated date was passed over, the automatic backup is executed.
 When "-1" is set to "Auto backup day 1", the automatic backup is executed every turning NC power ON.
 When "0" is set to all on "Auto backup day 1" to "4", the automatic backup is not executed. It is possible to specify the designated date up to 4 days for a month.

---Setting range---

-1 to 31
 ("-1" can be set for "Auto backup day 1" only.)

【#8916】 Auto backup day 2

When the NC power is ON after the designated date was passed over, the automatic backup is executed.
 When "-1" is set to "Auto backup day 1", the automatic backup is executed every turning NC power ON.
 When "0" is set to all on "Auto backup day 1" to "4", the automatic backup is not executed. It is possible to specify the designated date up to 4 days for a month.

---Setting range---

-1 to 31
 ("-1" can be set for "Auto backup day 1" only.)

【#8917】 Auto backup day 3

When the NC power is ON after the designated date was passed over, the automatic backup is executed.
 When "-1" is set to "Auto backup day 1", the automatic backup is executed every turning NC power ON.
 When "0" is set to all on "Auto backup day 1" to "4", the automatic backup is not executed. It is possible to specify the designated date up to 4 days for a month.

---Setting range---

-1 to 31
 ("-1" can be set for "Auto backup day 1" only.)

【#8918】 Auto backup day 4

When the NC power is ON after the designated date was passed over, the automatic backup is executed.
 When "-1" is set to "Auto backup day 1", the automatic backup is executed every turning NC power ON.
 When "0" is set to all on "Auto backup day 1" to "4", the automatic backup is not executed. It is possible to specify the designated date up to 4 days for a month.

---Setting range---

-1 to 31
 ("-1" can be set for "Auto backup day 1" only.)

【#8919】 Auto backup device

Select the automatic backup target device.
 *The setting range differs according to the model.

---Setting range---

[M700 Series]
 0: DS
 1: HD
 2: Memory card

[M70 Series]
 0: Memory card

【#8920】 3D tool ofs select

Select the method to calculate the drawing position when drawing a solid.
With 3D drawing, the drawing position (tool tip position) is calculated with the method designated with this parameter, and the image is drawn.

0: Machine position ± tool shape setting window data

1: Machine position ± tool compensation amount

2: Machine position ± tool shape setting window data

3: Machine position ± tool shape setting window data

【#8921】 Mass Edit select

Select the editing mode for the machining programs saved in HD, FD, and memory card.

When the program size is 1.0MB (When "#8910 Edit Undo" is invalid, 2.0MB) or more, mass-editing will be applied.

0: Regular editing mode

1: Mass-editing mode

【#8922】 T-reg-dup check

Set whether to enable the duplication check in registering tools to magazine pots, and in setting tool Nos. for spindle/standby.

0: Duplication check valid for all valid magazines

1: Duplication check invalid

2: Duplication check valid only for the selected magazine

【#8923(PR)】 Hide Edit-IO menu

Set whether to enable the edit-in/out menu.

When disabled, the edit-input/output menu won't appear.

However, the maintenance-in/out menu is always enabled regardless of this parameter setting.

0: Enable

1: Disable

【#8924】 MEAS. CONFIRM MSG

Select whether to display a confirming message when attempting to write compensation data for tool measurement, or coordinate system data for workpiece measurement.

0: Not display a confirming message

1: Display a confirming message

【#8925】 SP on 1st part sys

Set a spindle No. to be displayed on the 1st part system window when 2-part system simultaneous display is valid.

High-order: Select an upper side spindle No.

Low-order: Select a lower side spindle No.

(Note 1) When "00" is set, spindles will be displayed in a default order (the 1st spindle on the upper side, the 2nd spindle on the lower side)

(Note 2) If you designate a bigger number than the setting of "#1039 spinno", or either the high-order or low-order setting is "0", the 1st spindle will be displayed.

---Setting range---

High-order: 0 to 6

Low-order: 0 to 6

【#8926】 SP on 2nd part sys

Set a spindle No. to be displayed on the 2nd part system window when 2-part system simultaneous display is valid.

High-order: Select an upper side spindle No.

Low-order: Select a lower side spindle No.

(Note 1) When "00" is set, spindles will be displayed in a default order (the 1st spindle on the upper side, the 2nd spindle on the lower side)

(Note 2) If you designate a bigger number than the setting of "#1039 spinno", or either the high-order or low-order setting is "0", the 1st spindle will be displayed.

---Setting range---

High-order: 0 to 6

Low-order: 0 to 6

【#8927】 SP on 3rd part sys

Set a spindle No. to be displayed on the 3rd part system window when 2-part system simultaneous display is valid.

High-order: Select an upper side spindle No.

Low-order: Select a lower side spindle No.

(Note 1) When "00" is set, spindles will be displayed in a default order (the 1st spindle on the upper side, the 2nd spindle on the lower side)

(Note 2) If you designate a bigger number than the setting of "#1039 spinno", or either the high-order or low-order setting is "0", the 1st spindle will be displayed.

---Setting range---

High-order: 0 to 6

Low-order: 0 to 6

【#8928】 SP on 4th part sys

Set a spindle No. to be displayed on the 4th part system window when 2-part system simultaneous display is valid.

High-order: Select an upper side spindle No.

Low-order: Select a lower side spindle No.

(Note 1) When "00" is set, spindles will be displayed in a default order (the 1st spindle on the upper side, the 2nd spindle on the lower side)

(Note 2) If you designate a bigger number than the setting of "#1039 spinno", or either the high-order or low-order setting is "0", the 1st spindle will be displayed.

---Setting range---

High-order: 0 to 6

Low-order: 0 to 6

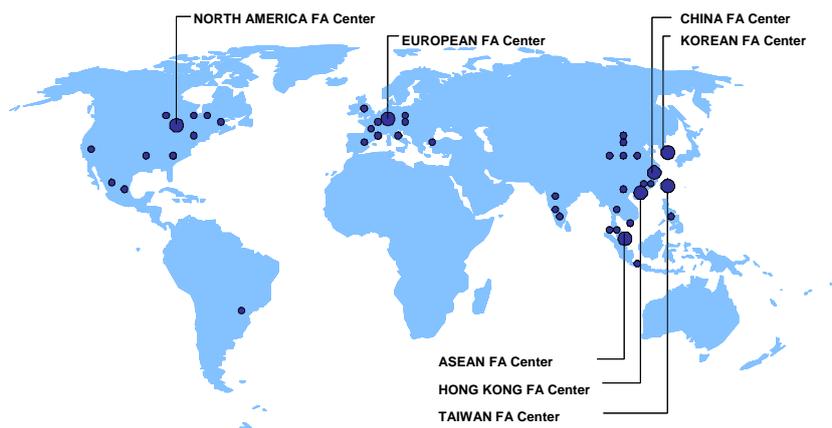
Revision History

Date of revision	Manual No.	Revision details
Aug. 2004	IB(NA)1500042-A	First edition created.
Mar. 2005	IB(NA)1500042-B	<ul style="list-style-type: none"> • Contents were revised to correspond to Mitsubishi CNC 700 Series software version B0. • The following sections were added to "I. SCREEN OPERATIONS": <ul style="list-style-type: none"> 1.1 Setting Display Unit Appearance 1.3 Screen Transition Diagram 1.4 Screen Selection Procedure 1.5 Setting Data 1.8 Guidance Function 2.2.2 Changing Whether to Show or Hide the Comment Field 2.3.5 Operation Sequence for Program Restart 2.5.8 Switching Full-screen Display Mode 2.16.2 Setting the Time Display Selection 2.22 Load Meter Display 2.23 Spindle, Standby Display 3.4.4 Erasing the Tool Registration Data 3.12 T Code List 4.2.17 G Code Guidance 4.3.8 Switching to Full-screen Display Mode 4.4.8 Switching to Full-screen Display Mode 5.7 Self Diagnosis Screen 5.8 Data Sampling Screen 6.3 All Backup Screen 6.4 System Setup Screen 6.5 Adjust S-analog Screen 6.6 Absolute Position Setting Screen 6.7 Auxiliary Axis Test Screen 6.8 Diagnosis Data Collection Setting Screen • The following section names were changed: <ul style="list-style-type: none"> "2.3.3 Directory change screen" -> "2.3.3 File Setting Screen" "5.1 Hardware and software configuration screen (H/W S/W config screen)" -> "5.1 System Configuration Screen " • The following sections were unified. <ul style="list-style-type: none"> "4.5.9 Formatting an FLD" and "4.5.10 Formatting a Memory card and DS" -> "4.5.9 Formatting an External Device " "6.2.9 Formatting an FLD" and "6.2.10 Formatting a Memory card and DS" -> "6.2.9 Formatting an External Device " • The configuration of "1. Operating the Setting and Display Unit" in "I. SCREEN OPERATIONS" was changed. • The following sections were added to " II. MACHINE OPERATIONS": <ul style="list-style-type: none"> 7. Stored Stroke Limit • "III. MAINTENANCE" was added. • The following sections were added to " IV. APPENDIXES": <ul style="list-style-type: none"> 6.2.15 System setup-related operation messages 6.2.16 Automatic backup-related operation messages Appendix 7. G Code Guidance Display List • Mistakes were corrected.

Date of revision	Manual No.	Revision details
Sept. 2005	IB(NA)1500042-C	<ul style="list-style-type: none"> • Contents were revised to correspond to Mitsubishi CNC 700 Series software version B3. • The following sections were added to "I. SCREEN OPERATIONS": <ul style="list-style-type: none"> 1.8.2 Alarm Guidance 1.9 Touch Panel Functions 1.10 Touch Panel S/W Key 2.2.3 Changing the Sorting Method 3.3.1 Tool Measurement (M system) 3.3.2 Tool Measurement (L system) 3.7.4 Carrying Out Rotation Measurement 3.13 Pallet Program Registration 5.9 Anshin-net Screen • The following section was added to "IV. APPENDIXES": <ul style="list-style-type: none"> 6.2.17 Messages Related to Anshin-net • The following sections were changed to paragraph header: <ul style="list-style-type: none"> 3.3.1 Carrying Out Tool Length Measurement 3.3.2 Carrying Out Tool Radius Measurement • The following sections were deleted: <ul style="list-style-type: none"> 6.1.2 Selecting the Parameter No. 6.1.6 Machine Parameters • Mistakes were corrected.
Mar. 2006	IB(NA)1500042-D	<ul style="list-style-type: none"> • Contents were revised to correspond to Mitsubishi CNC 700 Series software version C0. • The following sections were added to "I. SCREEN OPERATIONS": <ul style="list-style-type: none"> 4.2.17 Adding Sequence No. (N No.) Automatically 4.2.19 Playback Editing 4.5.12 Program Display Lock C 5.6.1 Alarm History • Mistakes were corrected.
Sept. 2006	IB(NA)1500042-E	<ul style="list-style-type: none"> • Contents were revised to correspond to Mitsubishi CNC 700 Series software version D0. • The following sections were added to "I. SCREEN OPERATIONS": <ul style="list-style-type: none"> 1.5.2 Inputting Operations 1.11 Screen Saver (Backlight OFF) Function 2.25 All Spindles' Rotation Speed Display 3.7.2 Workpiece Measurement (L System) 4.5.14 Sharing Machining Data 5.9.4 Sharing Machining Data 5.10 Machine Tool Builder Network System (MTB net) Screen 6.2.13 Sharing Machining Data • The configuration of "3.2 Tool Measurement" was changed. • The configuration of "3.7 Workpiece Measurement" was changed because workpiece measurement (L system) is added. • Mistakes were corrected.

Date of revision	Manual No.	Revision details
Aug. 2007	IB(NA)1500042-F	<ul style="list-style-type: none"> • Contents were revised to correspond to Mitsubishi CNC 700/70 Series software version E1. • The following sections were added to "I. SCREEN OPERATIONS": <ul style="list-style-type: none"> 1.12 Screen Capture [70 Series Only] 1.13 Multi-part System Program Management 4.5.15 The Batch Input/Output the Machining Program of NC Memory 6.2.14 The Batch Input/Output the Machining Program of NC Memory • The following sections were added to "II MACHINE OPERATIONS": <ul style="list-style-type: none"> 6.17 Tool retract return • The following sections were added to "III. MAINTENANCE": <ul style="list-style-type: none"> 1.2.4 How to Replace the Protective Sheet on the Touch Panel 1.3.3 How to Replace the Protective Sheet on the Touch Panel • Mistakes were corrected.
Dec. 2007	IB(NA)1500042-G	<ul style="list-style-type: none"> • Contents were revised to correspond to Mitsubishi CNC 700/70 Series software version E2. • The following sections were added to "I. SCREEN OPERATIONS": <ul style="list-style-type: none"> 1.10.2 Automatic Display of S/W Keyboard [70 Series Only] 2.1.3 Operation of 2-part System Simultaneous Display • The following sections were added to "III. MAINTENANCE": <ul style="list-style-type: none"> 5.3 Spindle Override 6.4 Z Axis Cancel 6.11 Mirror Image 6.15 F 1-digit Feed 6.19 Each Axis Machine Lock 6.22 External Deceleration 6.23 Reference Position Retract 6.24 Spindle Orientation 7.2 Chuck Barrier/Tailstock Barrier (L System) 7.3 Computer Link B 7.4 Manual Synchronous Tapping • The following sections were added to "IV. APPENDIXES": <ul style="list-style-type: none"> Appendix 8. IP Address Resetting Procedure at Disabled Network Communication [700 Series Only] Appendix 9. User Parameter List (Moved from "I. SCREEN OPERATIONS") • Mistakes were corrected.

Global service network



North America FA Center (MITSUBISHI ELECTRIC AUTOMATION INC.)

Illinois CNC Service Center
500 CORPORATE WOODS PARKWAY, VERNON HILLS, IL. 60061, U.S.A.
TEL: +1-847-478-2500 (Se) FAX: +1-847-478-2650 (Se)

California CNC Service Center
5665 PLAZA DRIVE, CYPRESS, CA. 90630, U.S.A.
TEL: +1-714-220-4796 FAX: +1-714-229-3818

Georgia CNC Service Center
2810 PREMIERE PARKWAY SUITE 400, DULUTH, GA., 30097, U.S.A.
TEL: +1-678-258-4500 FAX: +1-678-258-4519

New Jersey CNC Service Center
200 COTTONTAIL LANE SOMERSET, NJ. 08873, U.S.A.
TEL: +1-732-560-4500 FAX: +1-732-560-4531

Michigan CNC Service Satellite
2545 38TH STREET, ALLEGAN, MI., 49010, U.S.A.
TEL: +1-847-478-2500 FAX: +1-817-416-1439

Ohio CNC Service Satellite
62 W. 500 S., ANDERSON, IN., 46013, U.S.A.
TEL: +1-847-478-2608 FAX: +1-847-478-2690

Texas CNC Service Satellite
1000, NOLEN DRIVE SUITE 200, GRAPEVINE, TX. 76051, U.S.A.
TEL: +1-817-251-7468 FAX: +1-817-416-1439

Canada CNC Service Center
4299 14TH AVENUE MARKHAM, ON. L3R 0J2, CANADA
TEL: +1-905-475-7728 FAX: +1-905-475-7935

Mexico CNC Service Center
MARIANO ESCOBEDO 69 TLALNEPANTLA, 54030 EDO. DE MEXICO
TEL: +52-55-9171-7662 FAX: +52-55-9171-7698

Monterrey CNC Service Satellite
ARGENTINA 3900, FRACC. LAS TORRES, MONTERREY, N.L., 64720, MEXICO
TEL: +52-81-8365-4171 FAX: +52-81-8365-4171

Brazil MITSUBISHI CNC Agent Service Center (AUTOMOTION IND. COM. IMP. E EXP. LTDA.)
ACESSO JOSE SARTORELLI, KM 2,1 18550-000 BOITUVA - SP, BRAZIL
TEL: +55-15-3363-9900 FAX: +55-15-3363-9911

European FA Center (MITSUBISHI ELECTRIC EUROPE B.V.)

Germany CNC Service Center
GOTHAER STRASSE 8, 40880 RATINGEN, GERMANY
TEL: +49-2102-486-0 FAX: +49-2102-486-591

South Germany CNC Service Center
KURZE STRASSE, 40, 70794 FILDERSSTADT-BONLANDEN, GERMANY
TEL: +49-711-3270-010 FAX: +49-711-3270-0141

France CNC Service Center
25, BOULEVARD DES BOUVETS, 92741 NANTERRE CEDEX FRANCE
TEL: +33-1-41-02-83-13 FAX: +33-1-49-01-07-25

Lyon CNC Service Satellite

U.K CNC Service Center
TRAVELLERS LANE, HATFIELD, HERTFORDSHIRE, AL10 8XB, U.K.
TEL: +44-1707-282-846 FAX: +44-1707-278-992

Italy CNC Service Center
ZONA INDUSTRIALE VIA ARCHIMEDE 35 20041 AGRATE BRIANZA, MILANO ITALY
TEL: +39-039-60531-342 FAX: +39-039-6053-206

Spain CNC Service Satellite
CTRA. DE RUBI, 76-80 -APDO.420 08190 SAINT CUGAT DEL VALLES, BARCELONA SPAIN
TEL: +34-935-65-2236 FAX:

Turkey MITSUBISHI CNC Agent Service Center (GENEL TEKNİK SİSTEMLER LTD. STI.)
DARULACEZE CAD, FAMAŞ IS MERKEZİ A BLOCK NO.43 KAT2 80270 OKMEYDANI
ISTANBUL, TURKEY
TEL: +90-212-320-1640 FAX: +90-212-320-1649

Poland MITSUBISHI CNC Agent Service Center (MPL Technology Sp. z. o. o.)
UL. SLICZNA 34, 31-444 KRAKÓW, POLAND
TEL: +48-12-632-28-85 FAX:

Wroclaw MITSUBISHI CNC Agent Service Satellite (MPL Technology Sp. z. o. o.)
UL. KOBIERZYCKA 23, 52-315 WROCLAW, POLAND
TEL: +48-71-333-77-53 FAX: +48-71-333-77-53

Czech MITSUBISHI CNC Agent Service Center (AUTOCONT CONTROL SYSTEM S.R.O.)
NEMOCNICNI 12, 702 00 OSTRAVA 2 CZECH REPUBLIC
TEL: +420-596-152-426 FAX: +420-596-152-112

ASEAN FA Center (MITSUBISHI ELECTRIC ASIA PTE. LTD.)

Singapore CNC Service Center
307 ALEXANDRA ROAD #05-01/02 MITSUBISHI ELECTRIC BUILDING SINGAPORE 159943
TEL: +65-6473-2308 FAX: +65-6476-7439

Thailand MITSUBISHI CNC Agent Service Center (F. A. TECH CO., LTD)
898/19,20,21,22 S.V. CITY BUILDING OFFICE TOWER 1 FLOOR 12,14 RAMA III RD
BANGPONGPANG, YANNAWA, BANGKOK 10120, THAILAND
TEL: +66-2-682-6522 FAX: +66-2-682-6020

Malaysia MITSUBISHI CNC Agent Service Center (FLEXIBLE AUTOMATION SYSTEM SDN. BHD.)
60, JALAN USJ 10/1B 47620 UEP SUBANG JAYA SELANGOR DARUL EHSAN MALAYSIA
TEL: +60-3-5631-7605 FAX: +60-3-5631-7636

JOHOR MITSUBISHI CNC Agent Service Satellite (FLEXIBLE AUTOMATION SYSTEM SDN. BHD.)
NO. 16, JALAN SHAHBANDAR 1, TAMAN UNGKU TUN AMINAH, 81300 SKUDAI, JOHOR MALAYSIA
TEL: +60-7-557-8218 FAX: +60-7-557-3404

Indonesia MITSUBISHI CNC Agent Service Center (PT. AUTOTEKNINDO SUMBER MAKMUR)
WISMA NUSANTARA 14TH FLOOR JL. M.H. THAMRIN 59, JAKARTA 10350 INDONESIA
TEL: +62-21-3917-144 FAX: +62-21-3917-164

India MITSUBISHI CNC Agent Service Center (MESSUNG SALES & SERVICES PVT. LTD.)
B-36FF, PAVANA INDUSTRIAL PREMISES M.I.D.C., BHOASRI PUNE 411026, INDIA
TEL: +91-20-2711-9484 FAX: +91-20-2712-8115

BANGALORE MITSUBISHI CNC Agent Service Satellite (MESSUNG SALES & SERVICES PVT. LTD.)
S 615, 6TH FLOOR, MANIPAL CENTER, BANGALORE 560001, INDIA
TEL: +91-80-509-2119 FAX: +91-80-532-0480

Delhi MITSUBISHI CNC Agent Parts Center (MESSUNG SALES & SERVICES PVT. LTD.)
1197, SECTOR 15 PART-2, OFF DELHI-JAIPUR HIGHWAY BEHIND 32ND MILESTONE
GURGAON 122001, INDIA
TEL: +91-98-1024-8895 FAX:

Philippines MITSUBISHI CNC Agent Service Center (FLEXIBLE AUTOMATION SYSTEM CORPORATION)
UNIT No.411, ALABANG CORPORATE CENTER KM 25, WEST SERVICE ROAD SOUTH
SUPERHIGHWAY, ALABANG MUNTINLUPA METRO MANILA, PHILIPPINES 1771
TEL: +63-2-807-2416 FAX: +63-2-807-2417

Vietnam MITSUBISHI CNC Agent Service Center (SA GIANG TECHNO CO., LTD)
47-49 HOANG SA ST. DAKAO WARD, DIST. 1 HO CHI MINH CITY, VIETNAM
TEL: +84-8-910-4763 FAX: +84-8-910-2593

China FA Center (MITSUBISHI ELECTRIC AUTOMATION (SHANGHAI) LTD.)

China CNC Service Center
2/F., BLOCK 5 BLDG. AUTOMATION INSTRUMENTATION PLAZA, 103 CAOBANG RD. SHANGHAI
200233, CHINA
TEL: +86-21-6120-0808 FAX: +86-21-6494-0178

Shenyang CNC Service Center
TEL: +86-24-2397-0184 FAX: +86-24-2397-0185

Beijing CNC Service Satellite
9/F, OFFICE TOWER 1, HENDERSON CENTER, 18 JIANGUOMENNEI DAJIE, DONGCHENG
DISTRICT, BEIJING 100005, CHINA
TEL: +86-10-6518-8830 FAX: +86-10-6518-8030

China MITSUBISHI CNC Agent Service Center (BEIJING JIAYOU HIGHTECH TECHNOLOGY DEVELOPMENT CO.)
RM 709, HIGH TECHNOLOGY BUILDING NO.229 NORTH SI HUAN ZHONG ROAD, HAIDIAN
DISTRICT, BEIJING 100083, CHINA
TEL: +86-10-8288-3030 FAX: +86-10-6518-8030

Tianjin CNC Service Satellite
RM909, TAIHONG TOWER, NO220 SHIZILIN STREET, HEBEI DISTRICT, TIANJIN, CHINA 300143
TEL: +86-22-2653-9090 FAX: +86-22-2635-9050

Shenzhen CNC Service Satellite
RM02, UNIT A, 13/F, TIANAN NATIONAL TOWER, RENMING SOUTH ROAD, SHENZHEN, CHINA
518005
TEL: +86-755-2515-6691 FAX: +86-755-8218-4776

Changchun Service Satellite
TEL: +86-431-50214546 FAX: +86-431-5021690

Hong Kong CNC Service Center
UNIT A, 25/F RYODEN INDUSTRIAL CENTRE, 26-38 TA CHUEN PING STREET, KWAI CHUNG, NEW
TERRITORIES, HONG KONG
TEL: +852-2619-8588 FAX: +852-2784-1323

Taiwan FA Center (MITSUBISHI ELECTRIC TAIWAN CO., LTD.)

Taichung CNC Service Center
NO.8-1, GONG YEH 16TH RD., TAICHUNG INDUSTRIAL PARK TAICHUNG CITY, TAIWAN R.O.C.
TEL: +886-4-2359-0688 FAX: +886-4-2359-0689

Taipei CNC Service Satellite
TEL: +886-4-2359-0688 FAX: +886-4-2359-0689

Tainan CNC Service Satellite
TEL: +886-4-2359-0688 FAX: +886-4-2359-0689

Korean FA Center (MITSUBISHI ELECTRIC AUTOMATION KOREA CO., LTD.)

Korea CNC Service Center
1480-6, GAYANG-DONG, GANGSEO-GU SEOUL 157-200, KOREA
TEL: +82-2-3660-9631 FAX: +82-2-3664-8668

Notice

Every effort has been made to keep up with software and hardware revisions in the contents described in this manual. However, please understand that in some unavoidable cases simultaneous revision is not possible.

Please contact your Mitsubishi Electric dealer with any questions or comments regarding the use of this product.

Duplication Prohibited

This manual may not be reproduced in any form, in part or in whole, without written permission from Mitsubishi Electric Corporation.

© 2004-2007 MITSUBISHI ELECTRIC CORPORATION
ALL RIGHTS RESERVED.

MITSUBISHI CNC



MODEL	700/70 Series
MODEL CODE	008—326
Manual No.	IB-1500042 (ENG)