MITSUBISHI ELECTRIC

Changes for the Better

Numerical Control (CNC)

Instruction Manual

M800V/M80V Series

IB-1501806(ENG)-L

Introduction

This manual describes the operations, production processes and maintenances for Mitsubishi Electric CNC.
Improper handling can cause unexpected malfunctions. To use this device correctly, be sure to read this manual before use.

Supported models of this manual are as follows:

Supported models	Abbreviations in this manual
M800VW Series	M850VW, M830VW
M800VS Series	M850VS, M830VS
M80VW Series	M80VW
M80V Series	M80V TypeA, M80V TypeB

Abbreviations for model names used in this manual are as follows:

Abbreviations	Supported models
M800V, M800V Series	M800VW Series/M800VS Series
M80V, M80V Series	M80VW Series/M80V Series
M800V/M80V, M800V/M80V Series	M800VW Series/M800VS Series/M80VW Series/M80V Series
M8V, M8V Series	M800VW Series/M800VS Series/M80VW Series/M80V Series

To safely use this CNC unit, thoroughly study the "Precautions for Safety" before use.
Be sure to keep this manual always at hand.
Also refer to the manuals on "Manual List" as necessary.

■Details described in this manual

- In this manual, the following abbreviations might be used.
  L system: Lathe system
  M system: Machining center system
  MTB: Machine tool builder
- In this manual, some "auxiliary axis" indicates PLC axis where axis indexing is performed by index position command.

Manual List

Manuals related to M800V/M80V Series are listed as follows.
These manuals are written on the assumption that all optional functions are added to the targeted model.
Some functions or screens may not be available depending on the machine or specifications set by MTB. (Confirm the specifications before use.)
The manuals issued by MTB take precedence over these manuals.

Manual	IB No.	Purpose and Contents
M800V/M80V Series Instruction Manual (1/2)	IB-1501806	♦ Explanation for display unit and screen operation ♦ Settings before machining, setup, etc.
M800V/M80V Series Instruction Manual (2/2)	IB-1501807	♦ Explanation for settings at machining, operation status, etc. ♦ Explanation for other functions
M800V/M80V Series Programming Manual (Lathe System) (1/2)	IB-1501619	♦ G code programming for lathe system ♦ Basic functions, etc.
M800V/M80V Series Programming Manual (Lathe System) (2/2)	IB-1501620	♦ G code programming for lathe system ♦ Functions for multi-part system, high-accuracy function, etc.
M800V/M80V Series Programming Manual (Machining Center System) (1/2)	IB-1501621	♦ G code programming for machining center system ♦ Basic functions, etc.
M800V/M80V Series Programming Manual (Machining Center System) (2/2)	IB-1501622	♦ G code programming for machining center system ♦ Functions for multi-part system, high-accuracy function, etc.
M800V/M80V Series Alarm Manual	IB-1501808	♦ Alarms
M800V/M80V Series Parameter Manual	IB-1501809	♦ Parameters

Manuals for MTBs (NC)

Manual	IB No.	Purpose and Contents
M800V/M80V Series Specifications Manual (Function)	IB-1501610	• Model selection • Outline of various functions
M800V/M80V Series Specifications Manual (Hardware)	IB-1501611	• Model selection • Specifications of hardware unit
M800VW/M80VW Series Connection and Setup Manual	IB-1501612	• Detailed specifications of hardware unit • Installation, connection, wiring, setup (startup/adjustment)
M800VS/M80V Series Connection and Setup Manual	IB-1501613	• Detailed specifications of hardware unit • Installation, connection, wiring, setup (startup/adjustment)
M800V/M80V Series PLC Development Manual	IB-1501614	• Electrical design • I/O relation (assignment, setting, connection), field network • Development environment (PLC on-board, peripheral development environment), etc.
M800V/M80V Series PLC Programming Manual (1/2)	IB-1501667	• Electrical design • Sequence programming • Explanation for instructions, functions, and parameters
M800V/M80V Series PLC Programming Manual (2/2)	IB-1501668	• Electrical design • Sequence programming • Usage examples of instructions
M800V/M80V Series PLC Interface Manual (1/2)	IB-1501804	• Electrical design • List of interface signals between NC and PLC
M800V/M80V Series PLC Interface Manual (2/2)	IB-1501805	• Electrical design • Details of interface signals between NC and PLC
M800V/M80V Series Maintenance Manual	IB-1501617	• Cleaning and replacement for each unit • Other items related to maintenance
NC Designer2 Instruction Manual	IB-1501250	• Explanation for handling NC NC Designer2
High Speed Processing Unit User's Manual	IB-1501714	• Specifications for high speed processing unit (HPU)
NC Machining AI Diagnosis NC MachiningAID Instruction Manual	IB-1501762	• Explanation for handling NC MachiningAID

Manuals for MTBs (drive section)

Manual	IB No.	Contents
MDS-E/EH Series Specifications Manual	IB-1501226	• Specifications for power supply regeneration type
MDS-E/EH Series Instruction Manual	IB-1501229	• Instruction for power supply regeneration type
MDS-EJ/EJH Series Specifications Manual	IB-1501232	• Specifications for regenerative resistor type
MDS-EJ/EJH Series Instruction Manual	IB-1501235	• Instruction for regenerative resistor type
MDS-EM/EMH Series Specifications Manual	IB-1501238	• Specifications for multi-hybrid, power supply regeneration type
MDS-EM/EMH Series Instruction Manual	IB-1501241	• Instruction for multi-hybrid, power supply regeneration type
DATA BOOK	IB-1501252	• Specifications of servo drive unit, spindle drive unit, motor, etc.
MDS-EX-CVP Series Specifications and Instruction Manual	IB-1501587	• Specifications and instruction for the power supply unit with large capacity

Manuals for MTBs (Others)

Manual	No.	Purpose and Contents
GOT2000 Series User's Manual (Hardware)	SH-081194ENG	• Outline of hardware such as part names, external dimensions, installation, wiring, maintenance, etc. of GOTs
GOT2000 Series User's Manual (Utility)	SH-081195ENG	• Outline of utilities such as screen display setting, operation method, etc. of GOTs
GOT2000 Series User's Manual (Monitor)	SH-081196ENG	• Outline of each monitor function of GOTs
GOT2000 Series Connection Manual (Mitsubishi Electric Products)	SH-081197ENG	• Outline of connection types and connection method between GOT and Mitsubishi Electric connection devices
GOT2000 Series Connection Manual (Non-Mitsubishi Electric Products 1)	SH-081198ENG	• Explanation for connection types and connection method between GOT and other company's devices
GOT2000 Series Connection Manual (Non-Mitsubishi Electric Products 2)	SH-081199ENG	
GOT2000 Series Connection Manual (Microcomputers, MODBUS/Fieldbus Products, Peripherals)	SH-081200ENG	• Explanation for connection types and connection method between GOT and microcomputers, MODBUS/fieldbus products, peripherals
GT SoftGOT2000 Version1 Operating Manual	SH-081201ENG	• Explanation for system configuration, screen configuration and operation method of monitoring software GT SoftGOT2000
GT Designer3 (GOT2000) Screen Design Manual	SH-081220ENG	• Outline of screen design method using screen creation software GT Designer3
GOT2000/GOT1000 Series CC-Link Communication Unit User's Manual	IB-0800351	• Explanation for handling CC-Link communication unit (for GOT2000 series/GOT1000 series)
GX Developer Version 8 Operating Manual (Startup)	SH-080372E	• Explanation for system configuration, installation, etc. of PLC development tool GX Developer
GX Developer Version 8 Operating Manual	SH-080373E	• Explanation for operations using PLC development tool GX Developer
GX Converter Version 1 Operating Manual	IB-0800004	• Explanation for operations using data conversion tool GX Converter
GX Works2 Installation Instructions	BCN-P5999-0944	• Explanation for the operating environment and installation method of GX Works2
GX Works2 Version 1 Operating Manual (Common)	SH-080779ENG	• Explanation for the system configuration of GX Works2 and the functions common to Simple project and Structured project such as parameter setting, operation method for the online function
GX Works2 Version 1 Operating Manual (Simple Project)	SH-080780ENG	• Explanation for methods for such as creating and monitoring programs in Simple project of GX Works2
GX Works2 Version 1 Operating Manual (Simple Project, Function Block)	SH-080984ENG	• Explanation for methods for such as creating function blocks, pasting function blocks to sequence programs, and operating FB library in Simple project of GX Works2
GX Works2 Version 1 Operating Manual (Structured Project)	SH-080781ENG	• Explanation for methods for such as creating and monitoring programs in Structured project of GX Works2
GX Works3 Installation Instructions	BCN-P5999-0391	• Explanation for the operating environment and installation method of GX Works3
MELSEC-Q/L/F Structured Programming Manual (Fundamentals)	SH-080782ENG	• Explanation for programming methods, types of programming languages, etc. required to create structured programs
MELSEC-Q/L Structured Programming Manual (Application Functions)	SH-080784ENG	• Explanation for specifications and functions related to application functions which can be used in structured programs
MELSEC-Q CC-Link System Master/ Local Module User's Manual	SH-080394E	• Explanation for system configuration, installation, wiring, etc. of master/local modules for CC-Link system
MELSEC iQ-R Programming Manual (CPU Module Instructions, Standard Functions/Function Blocks)	SH-081266ENG	• Explanation for instructions, general-purpose functions, and general-purpose function blocks required for programming the sequencer MELSEC iQ-R series

Precautions for Safety

Always read the specifications issued by the MTB, 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.

The following signs indicate prohibition and compulsory.

⃠	**This sign indicates prohibited behavior (must not do).** **For example, "Keep fire away" is indicated by ⃠.**
●	**This sign indicates a thing that is critical (must do).** **For example, "it must be grounded" is indicated by ⏚.**

The meaning of each pictorial sign is as follows.

⚠ **CAUTION**	⚠ **CAUTION rotate object**	⚠ **CAUTION HOT**	⚠ **Danger Electric shock risk**	⚠ **Danger explosive**
⃠ **Prohibited**	⃠ **Disassembly is prohibited**	⃠ **KEEP FIRE AWAY**	● **General instruction**	⏚ **Earth ground**

For Safe Use

Mitsubishi Electric CNC is designed and manufactured solely for applications to machine tools to be used for industrial purposes. Do not use this product in any applications other than those specified above, especially those which are substantially influential on the public interest or which are expected to have significant influence on human lives or properties.

⚠ DANGER

Not applicable in this manual.

⚠ WARNING

⚠ 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. Please confirm that G and F modal and coordinate values are appropriate. If there are coordinate system shift commands or M, S, T and B commands before the block set as the start position, carry out the required commands using the MDI, etc. If the program is run from the set block without carrying out these operations, there is a danger of interference with the machine or of machine operation at an unexpected speed, which may result in breakage of tools or machine tool or may cause damage to the operators.

⚠ Under the constant surface speed control (during G96 modal), if the axis targeted for the constant surface speed control moves toward the spindle center, the spindle rotation speed will increase and may exceed the allowable speed of the workpiece or chuck, etc. In this case, the workpiece, etc. may jump out during machining, which may result in breakage of tools or machine tool or may cause damage to the operators.

⚠ 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 (MTB) 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 the applicable functions are included. Some of them, however, may not be available for your NC system. Refer to the specifications issued by the machine tool builder before use.

⚠ Refer to the Instruction Manual issued by each MTB 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.

⚠ To protect the availability, integrity and confidentiality of the NC system against cyber-attacks including unauthorized access, denial-of-service (DoS) (*1) attack, and computer virus from external sources via a network, take security measures such as firewall, VPN, and anti-virus software.
(*1) Denial-of-service (DoS) refers to a type of cyber-attack that disrupts services by overloading the system or by exploiting a vulnerability of the system.

⚠ Mitsubishi Electric assumes no responsibility for any problems caused to the NC system by any type of cyber-attacks including DoS attack, unauthorized access and computer virus.

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 input/output device cable. Failure to do so could damage the input/output device and NC unit.

4. Items related to screen operation

⚠ Always keep the touchscreen surface clean. When it gets dirty, remove the dirt immediately. Do not perform any operation on the dirty touchscreen.

⚠ 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.

⚠ CAUTION

⚠ 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.

⚠ Selected machining surface is not canceled even when NC reset is executed, the coordinate system defined along the machining target surface (feature coordinate system) remains selected.
It depends on the setting of the parameter "#1247 set19/bit2" (Inclined surface machining mode hold) to hold or cancel the selected machining surface at the emergency stop state. However, selected machining surface is not held when the power is turned OFF.
Press the menu [Surface cancel] to cancel the selected machining surface. The operating coordinate system will be back to the machine coordinate system by canceling the machining surface.

⚠ The program including a character of any language other than the display language is not correctly displayed. Do not edit such a program. Any part of the program other than the comment part may also be changed if edited.

❗ 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 MTB.

5. Items related to programming

⚠ 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.

⚠ " ; ", "EOB", "%", and "EOR" are symbols used for explanation. The actual codes for ISO are "CR, LF" ("LF") and "%". "%" and "EOR" are represented by "%" in ISO.
The programs created on the edit screen are stored in the NC memory or NC memory 2 in a "CR, LF" format, however, the programs created with external devices 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 MTB.

6. Items related to operation

⚠ Do not enter the machine's movable range during automatic operation. During rotation, keep your 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 battery low warning is issued in the drive unit side, immediately replace the battery. Replace the batteries while applying the drive unit's control power.

❗ If the axis overruns or emits an abnormal noise, immediately press the emergency stop button and stop the axis movement.

⚠ CAUTION

Turn OFF the power immediately if any smoke, abnormal noise or odor is generated from the controller, drive unit or motor.

8. Items related to maintenance

Incorrect connections may damage the devices. Always connect the cables to the indicated connectors.

Do not apply voltages other than those indicated according to specification on the connector. Failure to observe this could cause bursting, damage, etc.

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-232C/USB interface, an electric shock or a unit failure may occur. It depends on the type of computer to be connected, or the use environment.
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.

⚠ Before using this product after a long period of storage, performance check must be conducted. Otherwise contact the MTB of your machine tool.

⚠ CAUTION

9. Other items

⚠ **Mitsubishi Electric CNC should be operated and stored under the following environment conditions. Refer to the instruction manual issued by the MTB for details of the environment conditions for the machine tool including the CNC unit.**

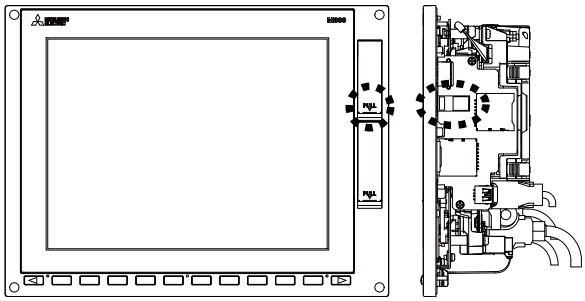
Unit name		Control unit		Display unit	Keyboard unit	Operation panel I/O unit
Ambient temperature	During operation	0 to 55 °C (M800VW/M80VW Series)	0 to 58 °C (M800VS/M80V Series)	0 to 58 °C		
	During storage	-20 to 60 °C				
Ambient humidity	Long term	10 to 75% RH (with no dew condensation)				
	Short term (*1)	10 to 95% RH (with no dew condensation)				
Vibration resistance		4.9 m/s^2 [0.5 G] or less				
Shock resistance		29.4 m/s^2 [3 G] or less				
Operating atmosphere		No corrosive gases, dust or oil mist				
Altitude		Operation/Storage: 1000 meters or less above sea level, Transportation: 13000 meters or less above sea level				

Note

(1) For the whole NC system, consider the characteristics of the drive units when the altitude is more than 1000 meters above sea level.
Refer to the manual of drive unit for details.

⚠ **Data transfer over wireless LAN may not be as stable as wired connection and packets might be lost depending on the surroundings and the location. Be sure to verify operation before using wireless LAN.** (*2)

⚠ **When the wireless LAN function is used, keep the wireless module at least 20 cm away from your body (head/torso). The wireless module is mounted on the part surrounded by the dotted line in the figure below.** (*2)



(*1) "Short term" means within one month.

(*2) M800VS/M80V Series

⚠ CAUTION

10. Notes on radio frequency interference (M800VS/M80V Series)

⚠ The wireless module in this product uses the radio channels in the same frequency band (channels 1 - 13 in the 2.4 GHz band) as the following equipment and radio stations:

- **Industrial, scientific and medical equipment**
- **Radio stations for RFID used for factory production lines**
  - **(1) Premises radio stations (radio stations requiring licenses)**
  - **(2) Specified low-power radio stations (radio stations not requiring licenses)**

Because using the wireless functionality could cause harmful interference to these equipment and radio stations, note the following:

- **Before using this product, ensure that no premises radio stations or specified low-power radio stations for RFID operate nearby.**
- **If this product causes harmful interference to premises radio stations for RFID, promptly change the frequency to avoid interference.**

Channels W53 and W56 in 5 GHz band are also used by weather radar and aerospace radar, and if there is radar activity on the channel, communication may be interrupted. You can avoid this problem by using channel W52 only, or enabling a function that immediately selects a different channel if radar is detected.
For information on how to configure these settings, refer to the manual for the wireless access point. The channels W52 and W53 in 5 GHz band are prohibited for outdoor use.

Other notes on using the wireless functionality are described on "15 Appendix 2: Wireless LAN Function [M800VS/M80V Series]". Read them in addition to the notes in this section.

Treatment of waste

The following two laws will apply when disposing of this product. Considerations must be made to each law. The following laws are in effect in Japan. Thus, when using this product overseas, the local laws will have a priority. If necessary, indicate or notify these laws to the final user of the product.

(1) Requirements for "Law for Promotion of Effective Utilization of Resources"
   (a) Recycle as much of this product as possible when finished with use.
   (b) When recycling, often parts are sorted into steel scraps and electric parts, etc., and sold to scrap contractors. Mitsubishi Electric recommends sorting the product and selling the members to appropriate contractors.

(2) Requirements for "Law for Treatment of Waste and Cleaning"
   (a) Mitsubishi Electric recommends recycling and selling the product when no longer needed according to item (1) above. The user should make an effort to reduce waste in this manner.
   (b) When disposing a product that cannot be resold, it shall be treated as a waste product.
   (c) The treatment of industrial waste must be commissioned to a licensed industrial waste treatment contractor, and appropriate measures, including a manifest control, must be taken.
   (d) Batteries correspond to "primary batteries", and must be disposed of according to local disposal laws.

Disposal



(Note) This symbol mark is for EU countries only.
This symbol mark is according to the directive 2006/66/EC Article 20 Information for end-users and Annex II.

Your MITSUBISHI ELECTRIC product is designed and manufactured with high quality materials and components which can be recycled and/or reused.
This symbol means that batteries and accumulators, at their end-of-life, should be disposed of separately from your household waste.
If a chemical symbol is printed beneath the symbol shown above, this chemical symbol means that the battery or accumulator contains a heavy metal at a certain concentration. This will be indicated as follows:
Hg: mercury (0.0005%), Cd: cadmium (0.002%), Pb: lead (0.004%)
In the European Union there are separate collection systems for used batteries and accumulators.
Please, dispose of batteries and accumulators correctly at your local community waste collection/recycling centre.

Please, help us to conserve the environment we live in!

Trademarks

MELDAS, MELSEC, EZSocket, EZMotion, iQ Platform, MELSEC iQ-R, MELSOFT, GOT, CC-Link, CC-Link/LT, CC-Link IE, CC-Link IE/field, CC-Link IE/field Basic, CC-Link IE TSN, EcoMonitorLight and SLMP are either trademarks or registered trademarks of Mitsubishi Electric Corporation in Japan and/or other countries.

Ethernet is a registered trademark of Xerox Corporation in the United States and/or other countries.
Microsoft®, Windows®, SQL Server®, Access® and Microsoft Edge are either trademarks or registered trademarks of Microsoft Corporation in the United States and/or other countries.
SD logo and SDHC logo are either registered trademarks or trademarks of SD-3C, LLC.
UNIX is a registered trademark of The Open Group in the United States and/or other countries.
Intel® and Pentium® are either trademarks or registered trademarks of Intel Corporation in the United States and/or other countries.
MODBUS® is either a trademark or a registered trademark of Schneider Electric USA, Inc. or the affiliated companies in Japan and/or other countries.
EtherNet/IP is a trademark of Open DeviceNet Vendor Association,Inc.
PROFIBUS-DP is either a trademark or a registered trademark of PROFIBUS User Organization.
Oracle® and JavaScript are either trademarks or registered trademarks of Oracle Corporation, the subsidiaries, or the affiliated companies in the United States and/or other countries.
VNC is a registered trademark of RealVNC Ltd. in the United States and other countries.
QR Code is a registered trademark of DENSO WAVE INCORPORATED.
TRELLIX is a registered trademark or a trademark of Musarubra US LLC or its subsidiaries in the United States and other countries.
Punchtap is licensed by EMUGE.
The term HDMI and the HDMI Logo are trademarks or registered trademarks of HDMI Licensing Administrator, Inc. in the United States and other countries.
Wi-Fi is either a trademark or a registered trademark of Wi-Fi Alliance.
BiSS is a registered trademark of iC-Haus GmbH.
SPRINT™ is a trademark of Renishaw plc.
Android and Google Chrome are either trademarks or registered trademarks of Google Inc.
iPadOS, macOS and Safari are trademarks of Apple Inc., registered in the U.S. and other countries and regions.

Other company and product names that appear in this manual are trademarks or registered trademarks of the respective companies.

本製品の取扱いについて

(日本語 /Japanese)
本製品は工業用 (クラス A) 電磁環境適合機器です。販売者あるいは使用者はこの点に注意し、住商業環境以外での使用をお願いいたします。

Handling of our product

(English)
This is a class A product. In a domestic environment this product may cause radio interference in which case the user may be required to take adequate measures.

본 제품의 취급에 대해서

(한국어 /Korean)
이 기기는 업무용 (A 급) 전자파적합기기로서 판매자 또는 사용자는 이 점을 주의하시기 바라며 가정외의 지역에 서 사용하는 것을 목적으로 합니다 .

Contents

Chapter 1 - 9 : Refer to Instruction Manual (1/2)

Chapter 10 and later : Refer to Instruction Manual (2/2)

1 Outline of Display Unit and Screens 1
1.1 Display Unit and Keyboard Unit 2
1.2 External Memory Interface 5
1.3 Standard Screen and Home Screen 6
1.4 Change Screen Buttons and Menu Buttons 8
1.4.1 Change Screen Buttons 8
1.4.2 Menu Operations 9
1.4.3 Menu List View Function 10
1.4.4 Main Menu List 12
1.4.5 Main Menu Functions 13

2 Screen Operations 21
2.1 Standard Screen 22
2.1.1 Screen Configuration 22
2.1.2 Displayed Part System 25
2.1.3 NC Status 26
2.1.4 Operation Mode/MDI Status 27
2.1.5 Operation Status 27
2.1.6 Alarms/Warnings 28
2.1.7 Operation Messages 28
2.1.8 Changing the Screen Theme Color 28
2.2 Home Screen 29
2.2.1 Screen Configuration 29
2.2.2 Switching Displayed Part System from the Operation Menu 34
2.2.3 Operating the Application 34
2.2.4 Displaying the Manuals [M800VW Windows-based Display Unit/M80VW Windows-based Display Unit] 36
2.3 Home Applications (19-type Vertical Display Unit) 41
2.3.1 Home Screen (Machine Status Display) 41
2.3.2 Keyboard Application 43
2.3.3 Operation Panel Application 44
2.3.4 Manual Application 45
2.3.5 Note Application 48
2.4 Setting Data 50
2.4.1 Setting Numerals and Alphabetical Characters 50
2.4.2 Inputting Operations 51
2.5 Guidance Function 53
2.5.1 Parameter Guidance 53
2.5.2 Alarm Guidance 55
2.5.3 G Code Guidance 57
2.6 Touchscreen Functions 60
2.6.1 Basic Operation 60
2.6.2 Software Keyboard 62
2.6.3 Touchscreen Operation Functions 67
2.6.3.1 Changing the Menu Display 68
2.6.3.2 Displaying the Menu Selection List 68
2.6.3.3 Scrolling the Displayed Data 69
2.6.3.4 Changing a Page 70
2.6.3.5 Switching to the Next Axis or Part System 71
2.6.3.6 Selecting a File 71
2.6.3.7 Displaying Edit Window/Selective Display Window 72
2.6.3.8 Moving a Window 73
2.6.3.9 Changing the Font Size 74
2.6.3.10 Displaying the Parameter Guidance 75
2.6.3.11 Displaying the Alarm Guidance 75
2.6.3.12 Enlarging or Reducing, Moving, and Rotating a 3D Check Shape 76
2.6.3.13 Displaying the Screen List 78
2.7 Menu Customization Function 79
2.8 Changing the Operation Level (Protect Setting Screen) 81
2.9 Assigning Arbitrary Character String to a User-defined Key 82
2.9.1 Format of Configuration File 83

2.9.2 Definition Types of User-defined Keys ... 84
2.9.2.1 Type 1 ... 84
2.9.2.2 Type 2 ... 86
2.9.3 Editing a Configuration File ... 88
2.10 Outputting the Screen Information (Screenshot Capture Function) [M800VS/M80V Series] ... 89

3 Inputting a Machining Program and Defining Variables ... 91
3.1 Inputting a Machining Program ... 92
3.2 Creating and Editing a Machining Program ... 92
3.3 Registering and Editing the MDI Program ... 104
3.4 Registering and Editing the Fixed Cycle Program ... 106
3.4.1 Fixed Cycle Operation Parameters ... 106
3.4.2 Transmitting and Erasing the Fixed Cycle Program ... 106
3.5 Erasing a File ... 107
3.6 Editing Operations ... 109
3.6.1 Changing the Display ... 109
3.6.2 Displaying an Arbitrary Line ... 112
3.6.3 Rewriting Data ... 112
3.6.4 Inserting Data (Initial Data Overwrite Mode) ... 113
3.6.5 Overwriting Data (Initial Data Insertion Mode) ... 114
3.6.6 Deleting Data ... 114
3.6.7 Searching for Character Strings ... 116
3.6.8 Replacing Character Strings ... 120
3.6.9 Copying and Pasting Data ... 121
3.6.10 Undoing Program Changes ... 123
3.6.11 Searching T codes ... 124
3.6.12 Searching a Program from Edit Screen ... 125
3.7 Program Editing Support Function ... 127
3.7.1 Displaying Input Mistakes ... 127
3.7.2 Performing Operation Search at the Cursor Position ... 132
3.7.3 Adding Sequence No. (N No.) Automatically ... 134
3.7.4 G Code Guidance ... 135
3.7.5 Playback Editing ... 136
3.7.6 Optimal program conversion ... 140
3.7.7 Displaying the Finished Shape ... 145
3.8 Editing a Multi-part System Program ... 146
3.8.1 Multi-part System Simultaneous Program Editing ... 146
3.8.2 Special Program Editing Display for Synchronization between Part Systems ... 153
3.9 Inserting Cycle Interactively ... 155
3.9.1 Types of Cycle ... 156
3.9.2 Name of Data ... 158
3.9.3 Screen Configuration ... 159
3.9.4 Operation Descriptions ... 160
3.9.5 Operation Procedure ... 162
3.9.6 Highlighting the Cycle Program ... 163
3.10 Setting Machining Conditions for Vibration Cutting Control ... 164
3.10.1 Screen Configuration ... 164
3.10.2 Operation Method ... 167
3.11 Process Analysis Function ... 170
3.11.1 Screen Configuration ... 170
3.11.2 Operation Method ... 178
3.12 Defining Variables ... 182
3.12.1 Common Variables ... 182
3.12.2 Local Variables ... 188

4 Checking a Machining Program on Screen ... 191
4.1 Graphic Check (2D) ... 192
4.1.1 Checking Continuously ... 199
4.1.2 Checking One Block at a Time ... 202
4.1.3 Setting the Coordinate System to Draw ... 203
4.1.4 Drawing during a Program Check ... 206
4.1.5 Changing the Display Range ... 206
4.1.6 Changing the Display Mode ... 208
4.1.7 Changing the Display Angle ... 210
4.1.8 Switching the Axis Movement Direction (+/-) ... 211
4.1.9 Displaying the Machining Program on Checking ... 212

4.1.10 Switching to Full-screen Display Mode 212
4.1.11 Setting Auto Graphic 213
4.1.12 Switching the Draw Path 215
4.1.13 Drawing a Path Corresponding to the Rotary Axis (L System) 216
4.1.14 Associating Graphic Check Images with a Machining Program 217
4.1.15 Setting the Drawing Speed of Program Check 218
4.1.16 Availability of Check Mode with Other Functions 219
4.1.17 Handling of Variable Commands, Programmable Parameter Input, and Programmable Compensation Data Input 221
4.1.18 Notes for Graphic Check (2D) 222
4.2 Graphic Check (3D) (M System) 227
4.2.1 Switching Check Method 234
4.2.2 Checking Continuously 234
4.2.3 Checking One Block at a Time 237
4.2.4 Checking Multiple Part Systems Simultaneously 238
4.2.5 Canceling the Program Check 239
4.2.6 Enlarging and Reducing the Workpiece Shape 239
4.2.7 Moving the Workpiece Shape 239
4.2.8 Rotating the Workpiece Shape 240
4.2.9 Performing an Interference Check 240
4.2.10 Setting the Workpiece Shape 241
4.2.11 Setting the Tool Shape 247
4.2.12 Associating Graphic Check Images with a Machining Program 247
4.2.13 Setting the Drawing Speed of Program Check 247
4.2.14 Drawing Travel Command of the Rotary Axis 248
4.2.15 Availability of Check Mode with Other Functions 251
4.2.16 Handling of Variable Commands, Programmable Parameter Input, and Programmable Compensation Data Input 254
4.2.17 Notes for Graphic Check (3D) 255
4.3 Graphic Check (3D) (L System) 262
4.3.1 Switching Check Method 270
4.3.2 Checking Continuously 270
4.3.3 Checking One Block at a Time 270
4.3.4 Checking Multiple Part Systems Simultaneously 270
4.3.5 Canceling the Program Check 270
4.3.6 Enlarging and Reducing the Workpiece Shape 270
4.3.7 Moving the Workpiece Shape 270
4.3.8 Rotating the Workpiece Shape 270
4.3.9 Performing an Interference Check 271
4.3.10 Setting the Workpiece Shape 272
4.3.11 Setting the Tool Shape 276
4.3.12 Checking a Machining Program between Part Systems 276
4.3.13 Associating Graphic Check Images with a Machining Program 277
4.3.14 Setting the Drawing Speed of Program Check 277
4.3.15 Availability of Check Mode with Other Functions 278
4.3.16 Handling of Variable Commands, Programmable Parameter Input, and Programmable Compensation Data Input 279
4.3.17 Notes for Graphic Check (3D) 279
4.4 3D Machining Simulation (M system) [M800VW Series] 285
4.4.1 Switching Check Method 290
4.4.2 Checking Continuously 290
4.4.3 Checking Block by Block 290
4.4.4 Checking Multiple Part Systems Simultaneously 290
4.4.5 Canceling the Program Check 290
4.4.6 Enlarging/Reducing the Workpiece Shape 290
4.4.7 Moving the Workpiece Shape 290
4.4.8 Rotating the Workpiece Shape 290
4.4.9 Performing an Interference Check 291
4.4.10 Setting the Workpiece Shape 292
4.4.11 Setting the Tool Shape 292
4.4.12 Associating Graphic Check Images with a Machining Program 292
4.4.13 Switching Simulation Modes 293
4.4.14 Availability of Check Mode with Other Functions 294
4.4.15 Handling of Variable Commands, Programmable Parameter Input, and Programmable Compensation Data Input 296
4.4.16 Precautions for 3D Machining Simulation 296

4.5 Checking the Tool Command (T Code List) ... 302
4.5.1 Searching T List ... 303

5 Inputting and Outputting a Machining Program ... 307
5.1 Inputting and Outputting a Program ... 308
5.2 Selecting a Device, Directory and File ... 314
5.3 Transferring a File ... 319
5.4 Comparing Files (Compare) ... 322
5.5 Erasing a File ... 323
5.6 Changing a File Name (Rename) ... 324
5.7 Creating a Directory ... 325
5.8 Other Functions ... 326
5.8.1 Merging a File ... 326
5.8.2 List of File Names ... 327
5.8.3 Edit Lock B and C ... 327
5.8.4 Program Display Lock ... 328
5.8.5 Data Protection Keys ... 329
5.8.6 Leading Zero ... 329
5.9 The Batch Input/Output the Machining Program of NC Memory/NC Memory 2 ... 332
5.10 Formatting Memory Card/DS/USB Memory ... 336

6 Setting a Tool and a Workpiece ... 337
6.1 Setting a Tool ... 338
6.2 Registering a Tool (Tool Registration Screen) ... 338
6.2.1 Registering a Tool in the Magazine Pot ... 341
6.2.2 Setting the PLC Command ... 341
6.2.3 Setting and Erasing the Tool No. of Spindle/Standby Tools ... 342
6.2.4 Erasing the Tool Registration Data ... 342
6.3 Measuring a Tool (Tool Measurement Screen) ... 343
6.3.1 Tool Measurement (M System) ... 343
6.3.2 Tool Measurement (L System) ... 348
6.4 Setting Tool Compensation Amount (Tool Offset Screen) ... 363
6.4.1 Setting the Tool Compensation Data ... 370
6.4.2 Switching the Input Method for the Tool Compensation Data ... 373
6.4.3 Erasing the Tool Compensation Data ... 373
6.4.4 Copying and Pasting the Tool Compensation Data ... 374
6.4.5 Undoing Change of Tool Compensation Data ... 374
6.4.6 Disabling the Setting of Tool Compensation Amount ... 375
6.4.7 Measuring Tool Compensation Amount ... 376
6.4.8 Moving to designated Compensation Number ... 385
6.5 Setting the Tool Management Data (Tool Management Screen) ... 386
6.5.1 List of Display Items ... 388
6.5.2 Data Setting Range ... 395
6.5.3 Menu Configuration ... 402
6.5.4 Registering New Tool Management Data ... 408
6.5.5 Switching the Item Type of the Tool Management Data Detailed Portion ... 408
6.5.6 Displaying Data by Designating Tool Management Data No. ... 409
6.5.7 Displaying Data by Designating Tool No. ... 410
6.5.8 Sorting the Data by Tool No. ... 411
6.5.9 Deleting the Tool Management Data ... 412
6.5.10 Copying and Pasting the Tool Management Data ... 413
6.5.11 Specifying Display Format and Setting Range of Additional Information ... 416
6.6 Tool Life Management ... 418
6.6.1 Tool Life Management (M System) ... 419
6.6.1.1 Registering Tool Life Management Data on the Group Unit Display Screen ... 420
6.6.1.2 Operation Descriptions (Tool life management I/III) ... 427
6.6.1.3 Operation Descriptions (Tool life management II) ... 427
6.6.1.4 Registering the Tool Life Management Data Groups on the Group List Display Screen (Tool life management I/II) ... 429
6.6.2 Tool Life Management (L System) ... 431
6.6.2.1 Registering the Tool Life Management Data (Tool Life Management I) ... 432
6.6.2.2 Registering Tool Life Management Data on the Group Unit Display Screen (Tool Life Management II) ... 434
6.6.2.3 Operation Descriptions (Tool life management I) ... 438
6.6.2.4 Operation Descriptions (Tool life management II) ... 439

6.6.2.5 Registering the Tool Life Management Data Groups on the Group List Display Screen (Tool Life Management II) ... 441
6.6.3 Tool Life Prediction Function ... 442
6.6.4 Tool Life Management Set Allocation to Part Systems ... 443
6.7 Setting and Measuring a Workpiece ... 444
6.7.1 Workpiece Measurement (M System) ... 444
6.7.1.1 Carrying Out Surface Measurement ... 447
6.7.1.2 Carrying Out Hole Measurement ... 449
6.7.1.3 Carrying Out Width Measurement ... 450
6.7.1.4 Carrying Out Rotation Measurement ... 452
6.7.1.5 Performing Automatic Recontact When Contacting the Workpiece ... 455
6.7.2 Workpiece Measurement (L System) ... 456
6.7.3 Measuring Instrument I/F Function ... 462
6.8 Setting Workpiece Coordinate System Offset ... 463
6.8.1 Setting the Coordinate System Offset Amount ... 466
6.8.2 Erasing the Coordinate System Offset Amount ... 467
6.8.3 Setting the Workpiece Coordinate Origin ... 467
6.8.4 Changing the Coordinate System Display ... 469
6.8.5 Changing the Input Menus to Display ... 469
6.8.6 Setting the Offset Amount of Workpiece Installation Error ... 470
6.8.7 Setting the Offset Amount of Workpiece Position for Rotary Axis ... 473
6.9 Setting Workpiece Coordinate System Shift (L System) ... 475
6.9.1 Manually Setting the Workpiece Coordinate System Shift Value ... 479
6.9.2 Automatically Setting the Workpiece Coordinate System Shift Value ... 482
6.10 Measuring the Workpiece Installation Error Data ... 484
6.10.1 Screen Configuration ... 491
6.10.2 Setting the Measurement Condition ... 492
6.10.3 Performing Workpiece Installation Error Measurement ... 494
6.10.4 Precautions for Workpiece Installation Error Measurement ... 495
6.11 Designating the Input Range (Range Setting Screen) ... 498
6.11.1 Designating the Input Range of the Tool Compensation Amount ... 499
6.11.2 Designating the Input Range of the Workpiece Coordinate System Offset Amount ... 501
6.11.3 Designating the Input Range of the Workpiece Coordinate System Shift Amount ... 503

7 Setting Parameters ... 505
7.1 Setting User Parameters ... 506
7.1.1 Selecting the Parameter Number ... 509
7.1.2 Setting the Parameters ... 510
7.1.3 Copying and Pasting Parameters ... 511
7.1.4 Undoing the Parameter Change ... 512
7.1.5 Parameter Configuration ... 513
7.1.6 Echoback ... 513
7.1.7 Ethernet parameter ... 513
7.1.8 Setting VNC Server Connection Password ... 513
7.1.9 Precautions ... 513
7.2 Selecting High-accuracy Control Parameters (Machining Cond I Screen) ... 514
7.3 Adjusting High-Accuracy Control Parameters ... 520

8 Setting Machine Movable Area ... 525
8.1 Setting the Stored Stroke Limit ... 526
8.1.1 Stored Stroke Limit II ... 528
8.2 Chuck Barrier/Tailstock Barrier (L System) ... 530

9 Other Setups ... 535
9.1 Pallet Program Registration ... 536
9.1.1 Standard Pallet Registration ... 537
9.1.2 Pallet List Screen ... 540
9.1.3 Pallet Details Screen ... 542
9.2 R-Navi ... 545
9.2.1 Selecting where to Register a Machining Surface (Machining Surface List Screen) ... 546
9.2.1.1 Selecting Where to Register a Machining Surface ... 548
9.2.1.2 Changing the Surface Name ... 548
9.2.1.3 Switching the Workpiece ... 549
9.2.1.4 Selecting the Basic Coordinate System ... 549
9.2.1.5 Automatically Registering Top and Side Faces as Machining Surfaces ... 550
9.2.1.6 Switching the Coordinate System Direction of a Workpiece Graphic ... 552

9.2.2 Registering and Setting the Workpiece (on the Work Setting Screen) 553
9.2.2.1 Registering the Workpiece 555
9.2.2.2 Changing the Workpiece Name 557
9.2.2.3 Changing the Form of the Work 557
9.2.2.4 Changing the Workpiece Size 558
9.2.3 Setting the Details of the Surface (on the Surface Detail Screen) 559
9.2.3.1 Moving the Cursor 562
9.2.3.2 Setting the Surface Coordinate Origin (Feature Coordinate Origin) 562
9.2.3.3 Specifying the Surface Coordinate Axial Directions 563
9.2.3.4 Checking the Setting Value of Coordinate Axis 568
9.2.3.5 Changing the Combination of Axial Direction 568
9.2.3.6 Switching the Surface 568
9.2.3.7 Rotating the Axial Direction of Coordinate Axis 569
9.2.3.8 Default Settings of Designation Method and Axis Combination 570
9.2.4 Surface Selection 571
9.2.4.1 Selecting a Machining Target Surface 573
9.2.4.2 Indexing the Machining Target Surface 574
9.2.4.3 Canceling the Machining Target Surface 575
9.2.4.4 Switching the Machining Target Surface 576
9.2.4.5 Switching the Workpiece 576
9.2.5 Shifting the Feature Coordinate Origin 577
9.2.5.1 Shifting the Feature Coordinate Origin to the Tool Center Position 579
9.2.5.2 Shifting the Feature Coordinate Origin by the Specified Amount 580
9.2.5.3 Shifting the Tool Center Position to the Position Specified on Inclined Surface Coordinate 581
9.3 Integrated Time 583
9.3.1 Integrated Time Display 583
9.3.2 Setting the Integrated Time 584
9.3.3 Setting the Time Display Selection 585
9.4 Counter Setting 586
9.5 Origin Set, Origin Cancel 588
9.6 Manual Numerical Value Command 590
9.7 MDI Program Editing 592
9.8 PLC Switch 594
9.8.1 Turning PLC Switches ON/OFF 595
9.9 Thread Recutting Function 596
9.9.1 Storing the Thread Groove Position 597
9.9.2 Switching the Stored Thread Groove ON/OFF 598
9.9.3 Erasing the Stored Thread Groove Position 598
9.10 Email Notification Function to Operator 599
9.10.1 Setting for Email Transmission Environment 600
9.10.2 Email Transmission Test 605
9.10.3 Description of Email Transmission Function 606
9.10.4 Inputting and Outputting Data 606
9.11 Checking the Settings Change History 608
9.11.1 Changing the Data Type Displayed in the Settings Change History 612
9.11.2 Undoing Changes 612
9.11.3 Displaying the Parameter Guidance 612

10 Machining a Workpiece 613
10.1 Operation Search 614
10.1.1 Executing an Operation Search 619
10.1.2 Changing Whether to Show or Hide the Comment Field 622
10.1.3 Changing the Sorting Method 623
10.2 Checking the Movement under Machining (Monitor Screen) 624
10.2.1 Screen Layout 624
10.2.2 Switching the Display Format 639
10.2.3 Switching the Counter Types 640
10.2.4 Switching the Spindle No. to be Displayed on Counter 643
10.2.5 Operation of 2-part/3-part/4-part System Simultaneous Display 643
10.2.6 Counter All-axis Display 645
10.2.7 Modal Display 647
10.2.8 Program Tree Display 652
10.2.9 Integrated Time Display 653
10.2.10 Load Meter Display 654
10.2.11 Spindle, Standby Display 656
10.2.12 Tool Center Coordinate Display 658

10.2.13 All Spindles' Rotation Speed Display 660
10.3 Checking the Movement under Machining (Trace Screen) 661
10.3.1 Displaying the Trace 665
10.3.2 Displaying the Machine Position of the Tool Center Point Trace 666
10.3.3 Changing the Display Range 666
10.3.4 Changing the Display Mode 668
10.3.5 Changing the Display Angle 669
10.3.6 Displaying the Machining Program on Checking 670
10.3.7 Switching the Full-screen Display Mode 670
10.3.8 Switching the Axis Movement Direction (+/-) 670
10.3.9 Drawing a Path Corresponding to the Rotary Axis 670
10.3.10 Precautions for Tracing 670
10.4 3D Machine Interference Check (M System) [M800VW/M80VW Series] 671
10.4.1 3D Monitor Screen (Main Screen) 674
10.4.2 Registering a Tool Model 675
10.4.3 Setting the Shape Data (Jig/Workpiece) 676
10.4.3.1 Arranging the Shape Data 679
10.4.3.2 Entering an Inclination Angle (Block) 679
10.4.4 Measuring a Reference Point of Shape 680
10.4.5 Setting the Shape Group as the Interference Check Target 684
10.4.6 Positioning the Shape Part 685
10.4.7 Setting Whether to Show/Hide the Components 687
10.4.7.1 Parts Selection Screen 687
10.4.7.2 Display Setting Screen 688
10.4.8 Inputting/outputting the Shape Group Data 689
10.4.9 Machining the Workpiece During Interference Check 691
10.4.9.1 Manual Operation 691
10.4.9.2 Automatic Operation 691
10.4.9.3 Canceling the Machine Interference Alarm 692
10.4.9.4 Disabling the Interference Check 692
10.5 Real-time 3D Machine Interference Check (M System) [M800VW Series] 692
10.6 Interference Check III 694
10.6.1 Main Screen 695
10.6.2 Display Setting Screen 699
10.6.3 Operation Method 700
10.7 Restart Search 702
10.7.1 Main Screen 706
10.7.2 Top Search Screen 708
10.7.3 File Setting Screen 709
10.7.4 MSTB History Screen 710
10.7.5 Operation Sequence for Program Restart 711
10.7.6 Executing Restart Search (Restart Type 1) 714
10.7.7 Executing Restart Search (Restart Type 2) 715
10.7.8 Executing Restart Search (Restart Search for All Part Systems in Batch) 718
10.7.9 Returning to the Restart Position 719
10.7.10 Executing the MSTB Commands 721
10.8 Collation and Stop 722
10.9 Correcting the Machining Program 726
10.9.1 Program Editing 726
10.9.2 Buffer Correction 732
10.10 Chatter Suppression 736
10.10.1 Adjusting the Variation Conditions 737

11 Troubleshooting 741

11.1 Confirming the NC Message (Alarm Screen) 742
11.1.1 Alarm History 744
11.2 Checking Machine Status 746
11.2.1 System Configuration Screen 746
11.2.2 Option Display Screen 749
11.2.3 Drive Monitor Screen (Servo Unit) 750
11.2.4 Drive Monitor Screen (Spindle Unit) 755
11.2.5 Drive Monitor Screen (Power Supply Unit) 760
11.2.6 Drive Monitor Screen (Synchronous Error) 762
11.2.7 Clearing the Alarm History on Drive Monitor Screen 763
11.3 Diagnosis Screens 764
11.3.1 Self Diagnosis Screen 764

11.3.2 Servo Diagnosis Screen ... 769
11.3.3 Collecting the Data on the Diagnosis Data Collection Setting Screen ... 772
11.4 Advanced Diagnosis ... 775
11.4.1 Displaying and Setting the PLC Device Data (I/F Diagnosis Screen) ... 775
11.4.1.1 Displaying the PLC Device Data ... 777
11.4.1.2 Carrying Out Modal Output ... 778
11.4.1.3 Carrying Out One-shot Output ... 779
11.4.2 Writing and Reading the Data Using the NC Data Designation (NC Memory Diagnosis Screen) ... 780
11.4.3 Collecting the NC Data (Data Sampling Screen) ... 782
11.5 Safety Observation ... 783
11.5.1 Safety Observation Screen (Configuration Diagnosis) ... 783
11.5.2 Safety Observation Screen (Signal Monitor) ... 785
11.5.3 Safety Observation Screen (Drive Monitor) ... 787
11.5.4 Safety Observation Screen (Version Display) ... 790

12 Performing a Backup Operation ... 791
12.1 Selecting a Device, Directory and File ... 796
12.2 Transferring a File ... 804
12.3 Comparing Files (Compare) ... 808
12.4 Erasing a File ... 809
12.5 Changing a File Name (Rename) ... 810
12.6 Creating a Directory ... 811
12.7 Restrictions ... 812
12.8 Other Functions ... 814
12.8.1 Merging a File ... 814
12.8.2 List of File Names ... 815
12.8.3 Edit Lock B and C ... 817
12.8.4 Data Protection Keys ... 818
12.8.5 The Batch Input/Output the Machining Program of NC Memory ... 818
12.8.6 Format Memory Card/DS/USB Memory ... 822
12.9 Performing Batch Backups of NC Memory Data ... 823
12.9.1 All Backup ... 826
12.9.2 All Restoration ... 828

13 Daily Maintenance ... 831

14 Appendix 1: Rotation Center Error Measurement Function (M System) / Rotation Center Error Compensation Function ... 835
14.1 Rotation Center Error Measurement Function (M System) ... 837
14.1.1 Screen Configuration ... 839
14.1.2 Measuring the Rotation Center Error Data ... 841
14.2 Rotation Center Error Measurement Function ... 844

15 Appendix 2: Wireless LAN Function [M800VS/M80V Series] ... 849
15.1 Configuring the Wireless LAN ... 852
15.1.1 Configuring for Access Point (AP) Mode ... 852
15.1.2 Configuring for Station (STA) Mode ... 854
15.2 Wireless LAN Diagnosis Screen ... 856
15.2.1 Setting the Access Point (AP) Nearby as Connection Destination ... 859
15.3 Description of Advanced Functions ... 860
15.3.1 DHCP ... 860
15.3.2 ESSID Stealth ... 863
15.3.3 AP Isolation ... 863
15.3.4 Abg Mode ... 863
15.3.5 BSSID ... 863
15.4 Relationship with Other Functions ... 864
15.5 Troubleshooting ... 865
15.6 The Software License Authorized for This Function ... 866

16 Appendix 3: IP Filter Setting Function ... 867
16.1 Setting the IP Address Filter ... 868
16.2 Examples for Using IP Address Filter ... 870

17 Appendix 4: VNC Server Function ... 873
17.1 Environment Establishment (Preparation) ... 874

17.2 Displaying the NC Screen with VNC Client 875
17.2.1 Connecting VNC Client 875
17.2.2 Operating the NC Using the Keyboard 877
17.2.3 Imposing Restrictions to VNC Client Operations 877
17.3 Troubleshooting 878

18 Appendix 5: Web Access (NC Screen Remote Browsing) [M800VS/M80V Series] 881
18.1 Setting for Web Access 883
18.1.1 Setting Parameters 883
18.1.2 Setting the IP Address 883
18.2 Displaying the CNC Screen on Web Browser 884
18.2.1 Explanations of the Screens 885
18.2.2 Operation Method 886

19 Appendix 6: List of Function Codes 887

20 Appendix 7: RS-232C I/O Device Parameter Setting Examples 889

21 Appendix 8: Other Functions for Reference 891
21.1 F1-digit Feed 892

1

Outline of Display Unit and Screens

This chapter describes the common functions of the screen.

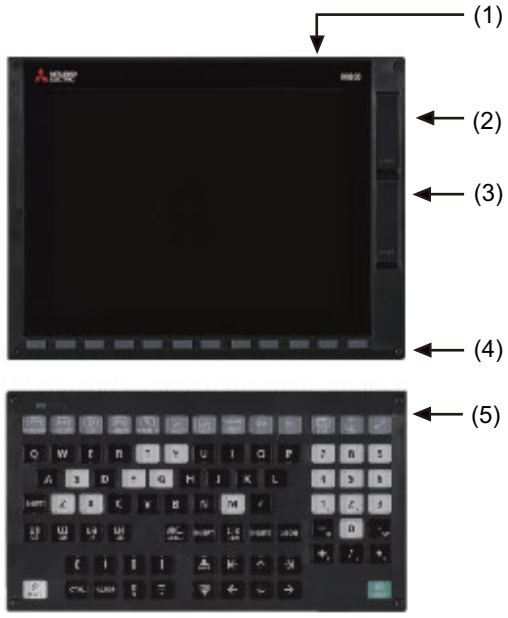
1.1 Display Unit and Keyboard Unit

Setting display unit consists of the display and keyboard unit. (Except for 19-type vertical display unit.)

For details on the 19-type vertical display unit, refer to "1.3 Standard Screen and Home Screen".

Names of parts are as follows.

■Example for 10.4-type display unit and vertical keyboard



(1) Display unit
(2) Front-side USB memory I/F
(3) Front-side SD card I/F
(4) Menu keys
(5) Keyboard unit

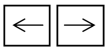
The following keys are provided on the menu key.

Key type	Key	Operation
Menu changeover keys	◁ (left side)	This cancels the menu item and returns to the upper level.
	▷ (right side)	This displays the menu items on the next page.
Menu keys		This opens the function screen for the selected menu item.

For details on menu operations, refer to "1.4.2 Menu Operations".

The keyboard contains the following keys.

Key type	Key	Operation
Function key (Function selection key)	MONITOR (MONITOR)	This displays the screen related to "operations".
	SET UP (SETUP)	This displays the screen related to "setup".
	EDIT (EDIT)	This displays the screen related to "editing".
	DIAGN (DIAGN)	This displays the screen related to "diagnosis".
	MAINTE (MAINTE)	This displays the screen related to "maintenance".
Page up/down keys (*1)	PAGE Page up 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.
	PAGE Page down 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.
Previous screen display key	BACK (BACK)	This redisplays the previously displayed screen.
Part system changeover key	$↔$ ($ <-> $)	When using a multi-part system, 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.
Data setting keys	A to Z, 0 to 9 Symbols such as +, -, =, /, :	These keys are used to set alphabetic characters, numerals, operation symbols, etc.
Particular keys	? Help key	This displays the operation guidance, parameter guidance and alarm guidance corresponding to the current operation.
		This action differs depending on MTB specifications.
	SFP F0	
Data correction keys	INSERT (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 (DELETE) Data delete key	This deletes the character just before the cursor position in the data setting area. When editing a program, the character on which the cursor is will be deleted and data after the cursor will move to the left.
	C•B CAN C·B: Clear Block key CAN: Cancel key	This cancels the setting in the data setting area.
Lower-case letter input key	ABC.../abc...	This changes the input between upper case and lower case alphabetic characters.
Block end key	;/EOB (EOB)	This inputs ";".
SHIFT key	SHIFT (SHIFT)	This validates the setting on the lower line of the data setting key.

Key type	Key	Operation
Cursor keys (*1)		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.
Frame keys		This switches the tab.
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.
RESET key	 (RESET)	This resets the NC.
Menu list key	 (LIST)	This is function that displays each screen's menu configuration as a list. For details, refer to "1.4.3 Menu List View Function".
Operation keys	 (ALTER)	This is alternate key (Alt key).
	 (CTRL)	This is control key.
	 (SP)	This is space key.
User-defined key		Arbitrary character string can be assigned to this key. Up to eight character strings can be registered. For details, refer to "2.9 Assigning Arbitrary Character String to a User-defined Key".

(*1) For M800VS/M80V Series, the behavior of the repeat key (page up/down key or cursor key) can be switched depending on the setting of the parameter "#19724 Repeat key switch".

0: Screen stop time depends on the length of time the button is pressed

1: The screen stops as soon as the button is released

For M800VW/M80VW Series, the repeat key behaves in the same manner as "1" (the screen stops as soon as the button is released).

1.2 External Memory Interface

An external memory interface is mounted on the right side of display unit. The interface accepts SD cards and USB memory.



Following operations are available for the stored data in the SD card or USB memory.

- Data input/output
- Machining program (text file) editing
- Machining program operation
- External search by PLC signal

The side memory interface unit is attached for 19-type LCD display unit. Refer to the MTB manual for the installation position.

Precautions

(1) We will not provide performance guarantee and maintenance for commercially available SD card or USB memory. If you use these media, careful performance check must be required.
Commercially available devices may not be compatible with devices made by Mitsubishi Electric, and they may not be suitable for FA environment temperature- or noise-wise.

(2) When inserting/removing an SD card, turn OFF the power of Mitsubishi Electric's device.

(3) When Inserting/removing a USB memory just after the device's power is ON, be sure to have enough interval to perform that (about 10 seconds or more).

(4) Machine vibration may cause the USB memory to fall out depending on environment. Therefore, the operation with the front-side USB memory is performed to handle on your own responsibility.

(5) Do not pull out the card or turn the power OFF while accessing the SD card or USB memory. Failure to observe this could cause the NC memory contents to be erased.
In case of emergency, always perform backups by having your important data duplicate, etc. as we will not guarantee the broken or lost data.
Refer to the manual of your SD card or USB memory for details.

(6) Do not connect a device other than a USB memory device. (including extension cable and USB hub, etc.)

(7) Keep the external memory interface cover closed except when in front-side USB memory mode.

(8) 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.

(9) Precautions for storage

- Do not store the card in locations having a high temperature or humidity.
- Do not store the card in dusty locations.

1.3 Standard Screen and Home Screen

The CNC screen can be switched between standard screen and home screen displays.

On the home screen, the spindle load, linear axis/rotary axis position, etc. can be displayed and machine status can be monitored. Also, each application can be called by pressing the application button on the home screen.
The applications registered to the home screen will differ depending on MTB.

The home screen display is configured using the parameter "#11080 HomeScreen display".

0: Not displayed

1: Displayed (home screen displayed when power is turned ON)

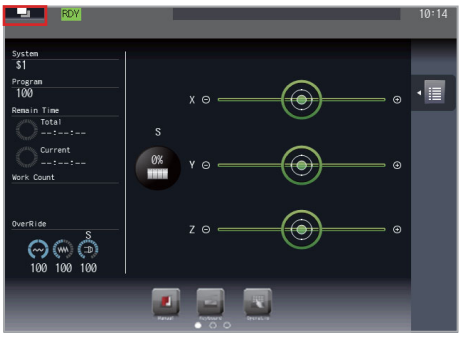
2: Displayed (home screen not displayed when power is turned ON)

For details on the standard screen, refer to "2.1 Standard Screen".
For details on the home screen, refer to "2.2 Home Screen".

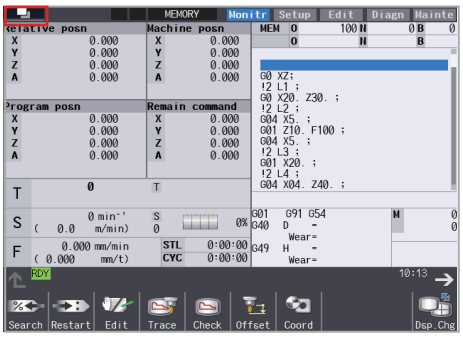
■Sample 10.4-type display unit

You can switch between the standard screen and home screen with the switch button on the upper left of the screen.

Home screen



Standard screen



The window display key ( ) can also be used to switch between screen display modes.
In home screen display mode, the screen can be changed to the specified screen directly by pressing a function key such as MONITOR or SETUP.

Note

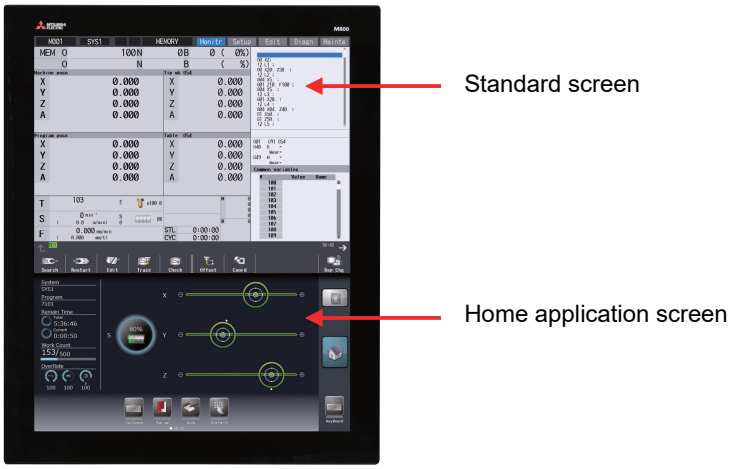
(1) When changing to the home screen with any window open, including the operation search window, the window is closed. Even after returning to the standard screen, the window will remain closed.

(2) The menu list or guidance does not appear while the home screen is displayed.

(3) When you move to the home screen with the software keyboard displayed, the keyboard is closed.

■For 19-type vertical display unit

The home application screen (expansion application screen) is displayed on the bottom of the standard screen.



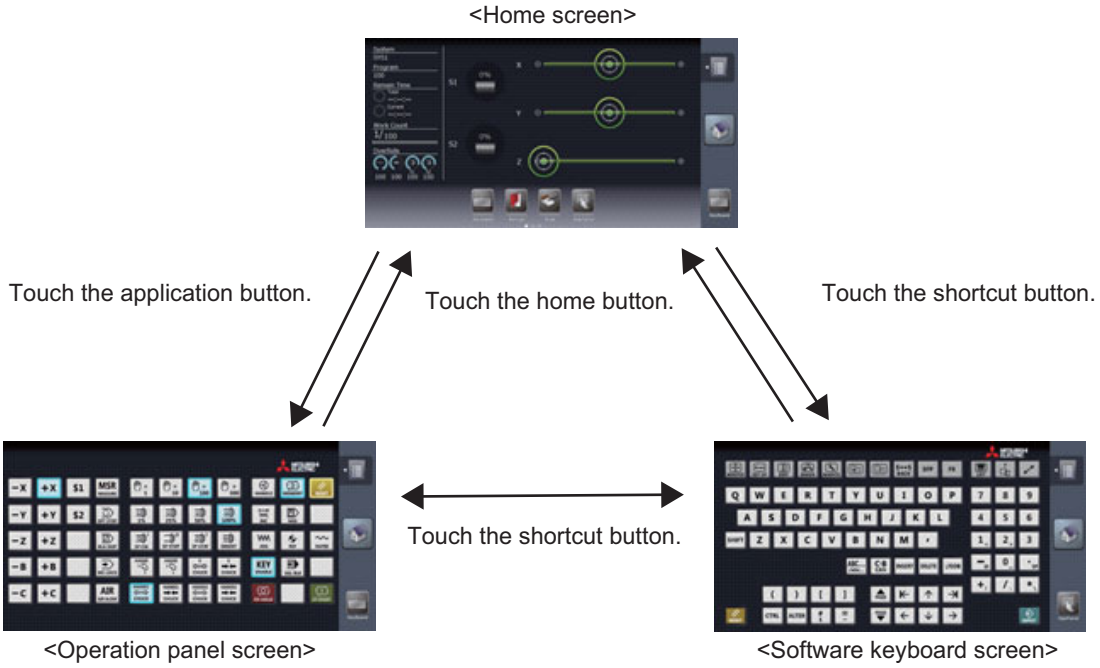
On the home application screen, the machine status, software keyboard, etc. will be displayed uncoupled from the upper half of the standard screen.

The following expansion applications are provided as standard with Mitsubishi Electric specifications:

- Home Screen (Machine Status Display)
- Keyboard Application
- Operation Panel Application
- Manual Application
- Note Application

MTB unique applications can also be added.

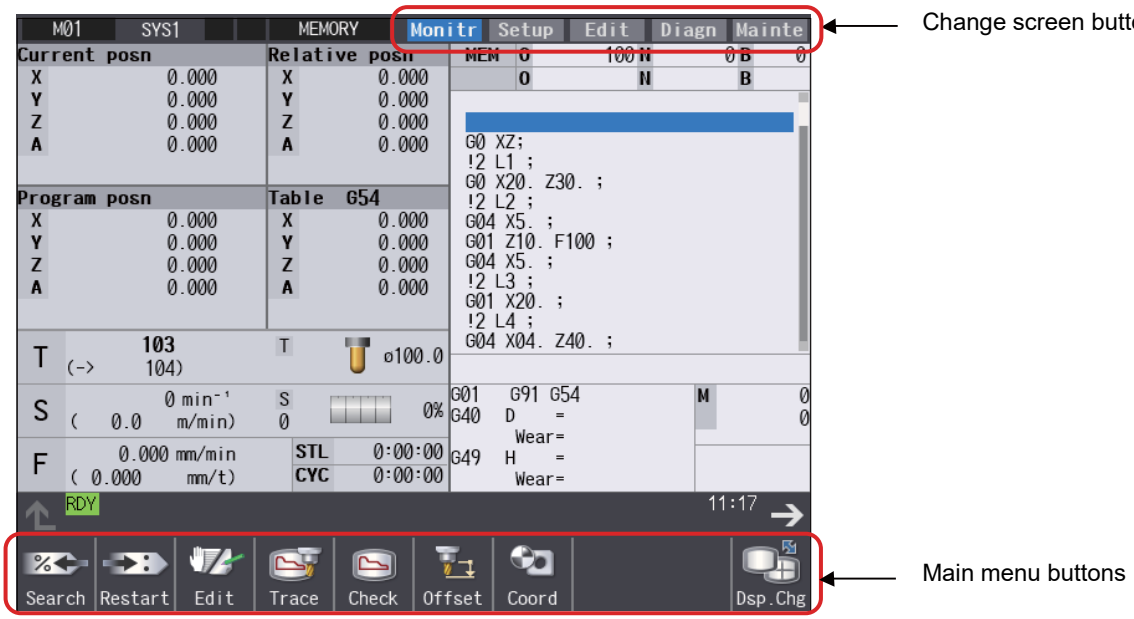
[Image of screen switching]



For details on home applications for the 19-type vertical display unit, refer to "2.3 Home Applications (19-type Vertical Display Unit)".

1.4 Change Screen Buttons and Menu Buttons

There are five change screen buttons in the upper left corner of the standard screen. Pressing the buttons will display the screen for their respective functions. The main menu for each screen is displayed at the bottom of the screen.



1.4.1 Change Screen Buttons

The change screen buttons serve the same function as the function keys on the keyboard.
For information on function keys, refer to "1.1 Display Unit and Keyboard Unit".

Change screen buttons	Function key (on keyboard)	Operation
Monitr	(MONITOR)	This displays the screen related to "operations".
Setup	(SETUP)	This displays the screen related to "setup".
Edit	(EDIT)	This displays the screen related to "editing".
Diagn	(DIAGN)	This displays the screen related to "diagnosis".
Mainte	(MAINTE)	This displays the screen related to "maintenance".

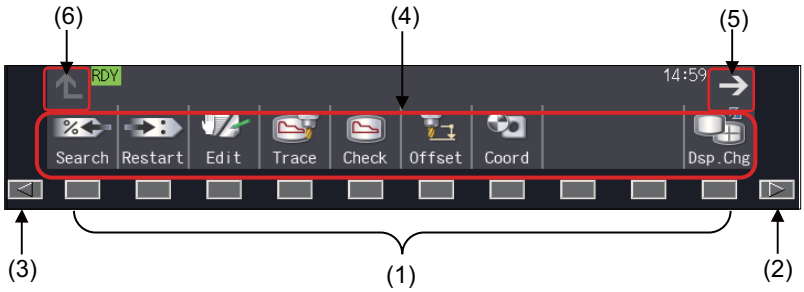
1.4.2 Menu Operations

The main menus contain buttons for switching between the functions handled on the operation, setup, and other screens. Icons are displayed on each main menu. Some menu icons differ between the M system and L system.

When the main menu is pressed, a detailed screen of the function will appear, along with buttons for operating that function. These buttons are referred to in this manual as sub-menu buttons.

Up to 10 menu buttons can be displayed on a single page.
Menu buttons can be operated by either the keyboard or touchscreen.

■Main menu and operation keys/buttons



■Sub-menu buttons with [Search] selected from the main menu



	Keyboard operation	Touchscreen operation	Details
Menu selection	Press one of the keys (1) below the menu buttons.	Press a menu button (4).	This opens the function screen for the selected menu button.
Display the next page of menu buttons (*1)	Press the ▷ key (2) on the right end.	Press the ⇨ button (5) on the screen.	When there are more than 10 menu buttons, this key displays menu buttons 11 and above.
Cancel menu	Press the ◁ key (3) on the left end.	Press the ⇦ button (6) on the screen.	If sub-menu buttons are displayed, this cancels the sub-menus and returns to the main menu for that screen.

(*1) Flick a menu button to the left (quickly running your fingertip horizontally along the screen) to slide and switch to the next menu. For details, refer to "2.6.3.1 Changing the Menu Display".

Note

(1) The 8.4-type display unit is not enabled for touchscreen support.

1.4.3 Menu List View Function

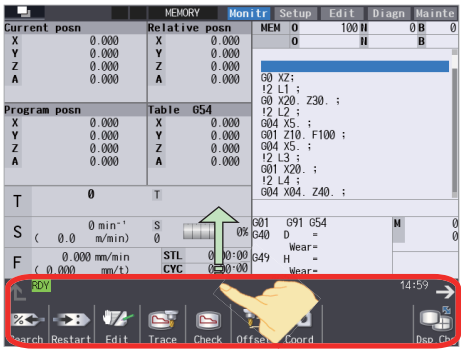
This function shows a list of the menu buttons at the top of the screen and then changes to the screen of your choice.

The following methods will display the menu list:

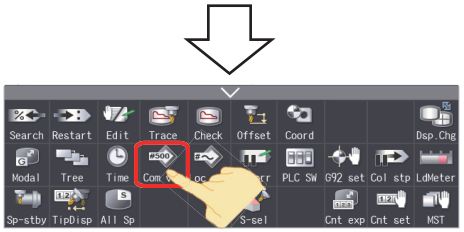
- Display the menu selection list by touch gestures
- Display the main menu list with the LIST key

Display the menu selection list

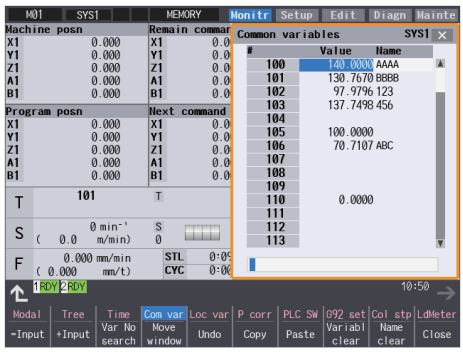
The menu selection list function shows a list of menu buttons (main menu/sub-menu buttons) for the currently selected screen and allows any menu to be selected directly.



Quickly run your fingertip across the screen to drag the screen upward.



Select a menu button from the menu list that appears.



The screen changes to the screen for the selected menu.

For details on menu selection list operations, refer to "2.6.3.2 Displaying the Menu Selection List".

Note

(1) If the selected screen has 10 or fewer menu buttons, no menu selection list appears.

Display the menu list screen

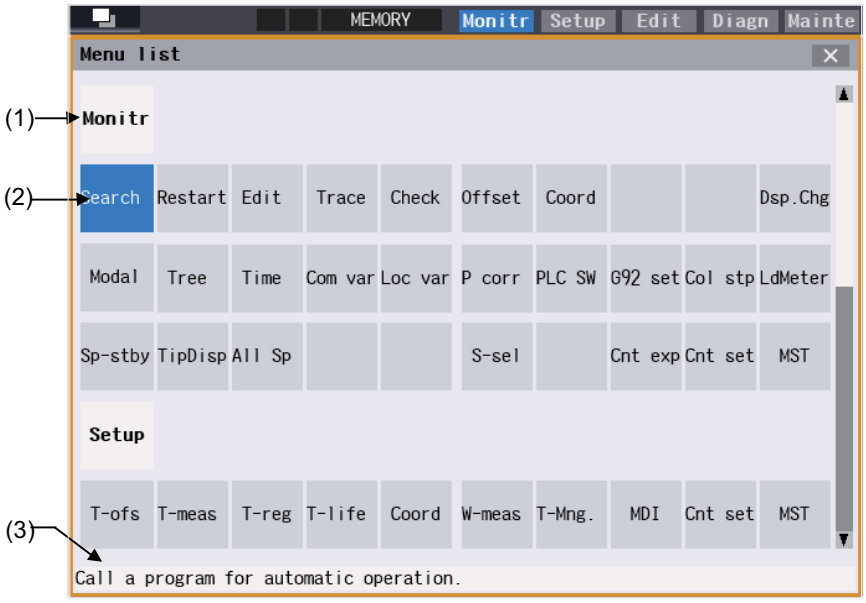
The menu list function displays each screen's main menu configuration as a list.

The menu list screen will display when the LIST key on the keyboard is pressed.

Select a menu button and a summary of the functions for that menu will display at the bottom of the screen.

To move directly to the operation screen for the selected menu, press the [INPUT] key.

Similarly, pressing the menu button a second time will change to the operation screen.



(1) The screen name is displayed.
Operation/Setup/Edit/Diagnosis/Maintenance

(2) The main menu list included on each screen is displayed.
[Search], [Restart], etc.

(3) A summary of functions for the currently selected main menu is displayed.

The menu list screen can be closed by:

- Pressing LIST again on the keyboard
- Pressing ◁ on the keyboard
- Pressing the ✕ button at the top left of the menu list window

1.4.4 Main Menu List

Each screen consists of the following main menu options.

The main menu options for the monitor (Monitr), setup (Setup), and edit (Edit) screens can be reorganized freely. For details, refer to "2.7 Menu Customization Function".

For an outline of main menu functions, refer to "1.4.5 Main Menu Functions".

Note

(1) Some menu icons are displayed differently for the M system and L system. The following is an example for 10.4-type and M system.

(2) The display contents differ depending on MTB settings.

■Monitor screen (Monitr)

Press the change screens button (Monitr) or the MONITOR function key to display this screen.

Page 1



Page 2



Page 3



Page 4



■Setup screen (Setup)

Press the change screens button (Setup) or the SET UP function key to display this screen.

Page 1



Page 2



Page 3



■Edit screen

Press the change screens button (Edit) or the EDIT function key to display this screen.

Page 1



■Diagnose screen (Diagn)

Press the change screens button (Diagn) or the DIAGN function key to display this screen.

Page 1



Page 2



■Maintenance screen (Mainte)

Press the change screens button (Mainte) or the MAINTE function key to display this screen.

Page 1

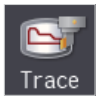


1.4.5 Main Menu Functions

The following is an overview of the main menus for each screen. Refer to the corresponding section for detailed operation.

The menus are the icons for 10.4-type.
In the table, "M" is used for M system icons and "L" is used for L system icons.

Monitor screen (Monitr)

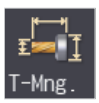
Menu button		Description	Reference
		This executes operation search.	10.1 Operation Search
		This executes restart search.	10.7 Restart Search
		This edits the searched machining program.	10.9 Correcting the Machining Program
M 	L 	This executes graphic trace.	10.3 Checking the Movement under Machining (Trace Screen)
		This checks the program.	4 Checking a Machining Program on Screen
M 	L 	This displays the tool compensation amount. A value can also be set for the tool compensation amount.	6.4 Setting Tool Compensation Amount (Tool Offset Screen)
		This displays the workpiece coordinate system offset. A value can also be set for the workpiece coordinate system offset.	6.8 Setting Workpiece Coordinate System Offset

Menu button		Description	Reference
		This displays the workpiece coordinate system shift. <Note> ♦This menu is not displayed for M system, or when the parameter "#11056 Workshift invalid" is set to "1".	6.9 Setting Workpiece Coordinate System Shift (L System)
		This displays the sub-menu to change the display type of the monitor screen.	10.2.2 Switching the Display Format
		This displays the program modal.	10.2.7 Modal Display
		This displays the program tree.	10.2.8 Program Tree Display
		This displays the date, time and integrated time, etc. The date, time and integrated time, etc., can also be set.	9.3 Integrated Time
		This displays the common variables. A value can also be set for the common variable.	3.12.1 Common Variables
		This displays the local variables.	3.12.2 Local Variables
		This corrects the buffer.	10.9.2 Buffer Correction
		This turns the PLC switches ON or OFF.	9.8 PLC Switch
		This sets and cancels the origin.	9.5 Origin Set, Origin Cancel
		This executes collation and stop.	10.8 Collation and Stop
		This displays the user PLC to display the spindle load, NC axis load, etc., with a bar graph. The display contents differ depending on MTB specification.	10.2.10 Load Meter Display
M 	L 	This displays the current spindle tool number and the tool number that is standby to be used next. The display contents differ depending on MTB specification.	10.2.11 Spindle, Standby Display

Menu button	Description	Reference
	Displays the tool tip coordinate, handle interruption amount (tool axis movement), and tool tip speed. <Note> ♦This menu is displayed in the following cases: - When any of the optional specifications (tool center point control, tool length compensation along the tool axis, tool handle feed & interruption, workpiece installation error compensation, inclined surface machining command, simple inclined surface machining command, 3-dimensional tool radius compensation (tool's vertical-direction compensation), 3-dimensional manual feed, or R-Navi) is valid - When the program format switch specification is valid	10.2.12 Tool Center Coordinate Display
	This displays the command rotation speed and actual rotation speed values of all spindles.	10.2.13 All Spindles' Rotation Speed Display
	This sets and inputs/outputs data related to the 3D machine interference check function or real-time 3D machine interference check function. Machine operation and interference part can be checked in the 3D model.	10.4 3D Machine Interference Check (M System) [M800VW/M80VW Series] 10.5 Real-time 3D Machine Interference Check (M System) [M800VW Series]
	This selects a machining surface to apply side machining or inclined surface machining to. <Note> ♦This menu is not displayed when the R-Navi additional specification is disabled.	9.2.4 Surface Selection
	This switches the counter display to replace the currently displayed axis.	10.2.4 Switching the Spindle No. to be Displayed on Counter
	This enlarges the counter to display all axes.	10.2.6 Counter All-axis Display
	An arbitrary value can be set in the relative position counter.	9.4 Counter Setting
	This executes a manual numerical value command.	9.6 Manual Numerical Value Command
	This sets the variation conditions for the chatter suppression control. <Note> ♦This menu is displayed for L system, and when the chatter suppression control specification is valid.	10.10 Chatter Suppression
	This displays the laser processing condition selection screen. <Note> ♦This menu is displayed for M system, and when the parameter "#11799 lsr_sys" (Laser processing control-enabled part system) is set to other than "0".	

Menu button	Description	Reference
	This displays the interference check III screen. <Note> •This menu is displayed when the interference check III specification is valid and the parameter "#1488 ITF3 valid" (Interference check III valid) is "1".	10.6 Interference Check III

Setup screen (Setup)

Menu button		Description	Reference
M 	L 	This sets tool compensation amount. There are three types of tool compensation: I, II, and III. The number of tool compensation sets to be designated or displayed varies depending on the optional specifications.	6.4 Setting Tool Compensation Amount (Tool Offset Screen)
M 	L 	This executes tool measurement. The distance from the basic point to the measurement point is measured by moving the tool to the measurement point manually, and this value can be set as the tool offset amount. <Note> •This menu is displayed when the parameter "#8932 Hide measure scrn" is set to "0".	6.3 Measuring a Tool (Tool Measurement Screen)
		This registers a tool. 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.	6.2 Registering a Tool (Tool Registration Screen)
M 	L 	This manages the tool life data. The life management data such as the tool usage is set and displayed. There are two types of the tool life management method.	6.6 Tool Life Management
		This sets the workpiece coordinate system offset. The offset value managed by NC can be set and displayed.	6.8 Setting Workpiece Coordinate System Offset
		This executes workpiece measurement, it also sets and measures the workpiece coordinate system shift. <Note> •This menu appears under the following conditions: - The parameter "#8932 Hide measure scrn" is set to "0". - The parameter "#11056 Workshift invalid" is set to "0" for L system.	6.7 Setting and Measuring a Workpiece 6.9 Setting Workpiece Coordinate System Shift (L System)
M 	L 	This sets and displays the management data for each tool. These data are relevant to which displayed on the [T-ofs] or [T-life] screen, that can be set the tool information. Thus the mutually-referenced data can be set and displayed on this screen.	6.5 Setting the Tool Management Data (Tool Management Screen)
		This edits the MDI program. The pop-up window appears to display the MDI program contents by pressing main menu [MDI].	9.7 MDI Program Editing

Menu button	Description	Reference
	This is to set the relative value counter. This displays the pop-up window of the relative position counter to execute the counter setting.	9.4 Counter Setting
	This sets an arbitrary value for S, M, T and B. Manual numerical value command is set and executed by inputting an address key, such as S, M, T and B.	9.6 Manual Numerical Value Command
M  L 	This searches the T command in designated machining program (including the subprograms) and lists it in the order of appearance.	4.5 Checking the Tool Command (T Code List)
	This registers the machining program per each pallet of the automatic pallet changer (hereinafter APC). <Note> •This menu is not displayed when the optional specification of pallet program registration is not added.	9.1 Pallet Program Registration
	This sets the user parameters. The parameter can be set and displayed by switching types.	7.1 Setting User Parameters
	This stores the thread groove position to enable re-thread machining. <Note> •This menu is not displayed when the optional specification of thread recutting is not added.	9.9 Thread Recutting Function
	This registers the workpiece and its target surface, on which side surface or inclined surface machining is executed. <Note> •This menu is not displayed for L system, or when the optional specification of R-Navi is not added.	9.2 R-Navi
	This sets the high-accuracy parameter which meets the machining purpose or machining process. <Note> •This menu is not displayed when the optional specifications related to high-accuracy control are not added.	7.2 Selecting High-accuracy Control Parameters (Machining Cond I Screen)
	This sets and displays the chuck barrier and tailstock barrier. <Note> •For M system, this menu is not displayed.	8.2 Chuck Barrier/Tailstock Barrier (L System)
	This measures the workpiece installation error and the measurement results are set in the workpiece coordinates system offset. <Note> •This menu is not displayed for L system, or when the optional specification of workpiece installation error compensation is not added.	6.10 Measuring the Workpiece Installation Error Data
	This sets environment/conditions of email transmission and destination address. <Note> •This menu is not displayed when the optional specification of email notification to operator is not added.	9.10 Email Notification Function to Operator

Menu button	Description	Reference
	This designates the input range (upper and lower limit) which can be set on each of the tool compensation screen, the workpiece coordinate system offset screen, and the workpiece coordinate system shift screen. <Note> •The menu name is [Set range] for the 15-type or 19-type display unit.	6.11 Designating the Input Range (Range Setting Screen)
	This lists the change history (alarm/warning history, tool offset change history, parameter change history, etc.). Setting values can also be restored to their original values by selecting the item from the list to be changed.	9.11 Checking the Settings Change History
	This displays the laser processing condition setting screen. <Note> •This menu is displayed for M system, and when the parameter "#11799 lsr_sys" (Laser processing control-enabled part system) is set to other than "0".	

Edit screen

Menu button	Description	Reference
	Edit the machining program.	3 Inputting a Machining Program and Defining Variables
	This checks the program. This function allows you to check the machining program without automatic operations. It can draw machining program movement path (2D), workpiece shape in the cutting process and tool movement (3D), and displays the operation result of NC internal data.	4 Checking a Machining Program on Screen
	This allows an operator to interactively set data to create a machining program automatically. The program can be created easily on the interactive pattern input screen. <Note> •This menu is not displayed when the parameter "#8996 Simple program ON" is set to "0".	
	This inputs and outputs the program between the NC internal memory and the external input/output device. <Note> •This menu is not displayed when the parameter "#8923 Hide Edit-IO menu" is set to "1".	5 Inputting and Outputting a Machining Program
	This analyzes the execution information of the machining program, and displays the analysis results. <Note> •This menu is displayed when the parameter "#11863 PrcssAnlysisSelect" (Process analysis selection) is "1".	3.11 Process Analysis Function

Diagnose screen (Diagn)

Menu button	Description	Reference
	This displays the hardware and software (software No. and version) configurations.	11.2.1 System Configuration Screen
	This displays the details of the optional specifications registered in the NC memory.	11.2.2 Option Display Screen
	This sets and displays input/output signals of the ladder program.	11.4.1 Displaying and Setting the PLC Device Data (I/F Diagnosis Screen)
	This displays drive diagnostic information (servo/spindle/power supply unit).	11.2.3 Drive Monitor Screen (Servo Unit) 11.2.4 Drive Monitor Screen (Spindle Unit) 11.2.5 Drive Monitor Screen (Power Supply Unit) 11.2.6 Drive Monitor Screen (Synchronous Error) 11.2.7 Clearing the Alarm History on Drive Monitor Screen
	This sets and displays NC internal data.	11.4.2 Writing and Reading the Data Using the NC Data Designation (NC Memory Diagnosis Screen)
	This displays a list of currently occurring alarms and their messages.	11.1 Confirming the NC Message (Alarm Screen)
	This displays the status of the hardware and operation stop.	11.3.1 Self Diagnosis Screen
	This displays the hardware and operation stop state.	11.4.3 Collecting the NC Data (Data Sampling Screen)
	This executes the diagnosis related to functional safety. <Note> ♦This menu button is displayed when the parameter "#1481 Enable S-Safety" (Enable smart safety observation) is set to "1".	11.5 Safety Observation
	This displays the operating status of wireless LAN and sets the connection access point (AP).	15.2 Wireless LAN Diagnosis Screen
	This makes settings to output waveform data to NC MachiningAID. <Note> ♦This menu is displayed when the parameter "#19250 NCAID con. valid" is set to "1". ♦This is displayed for M800VW/M80VW Series.	

Maintenance screen (Mainte)

Menu button	Description	Reference
	This performs NC memory format, absolute position parameter setting and maintenance data backup.	11.3.2 Servo Diagnosis Screen 11.3.3 Collecting the Data on the Diagnosis Data Collection Setting Screen
	This sets and displays user parameters and machine parameters.	7 Setting Parameters
	This inputs and outputs the various data between the NC memory and the external input/output device.	12 Performing a Backup Operation
	This displays guidance on the effects of adjustment procedures and related parameters to assist in adjusting the high-accuracy parameters.	7.3 Adjusting High-Accuracy Control Parameters
	This measures the servo control system characteristics and adjusts the parameters by vibrating the motor and measuring/analyzing its feedback.	

2

Screen Operations

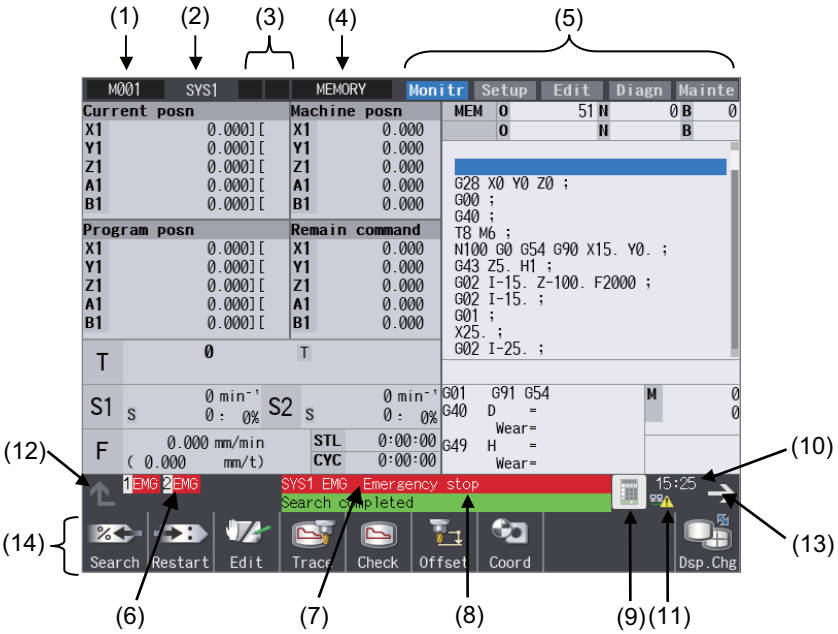
2.1 Standard Screen

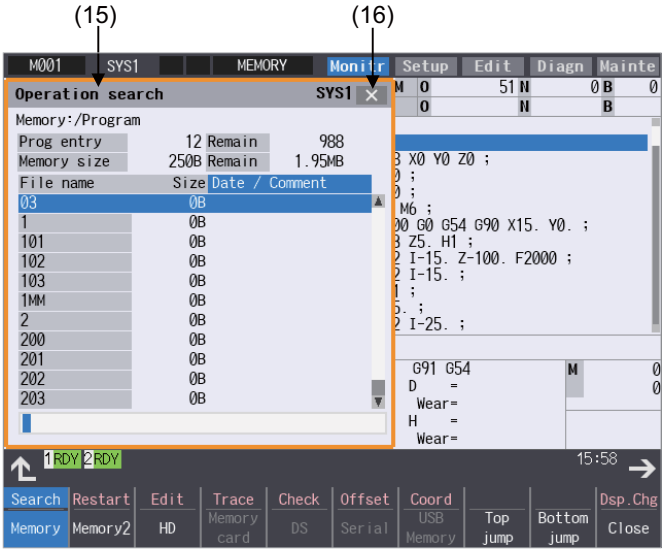
Note

(1) In this manual, a screen with no size indicated is intended for the 10.4-type display unit.

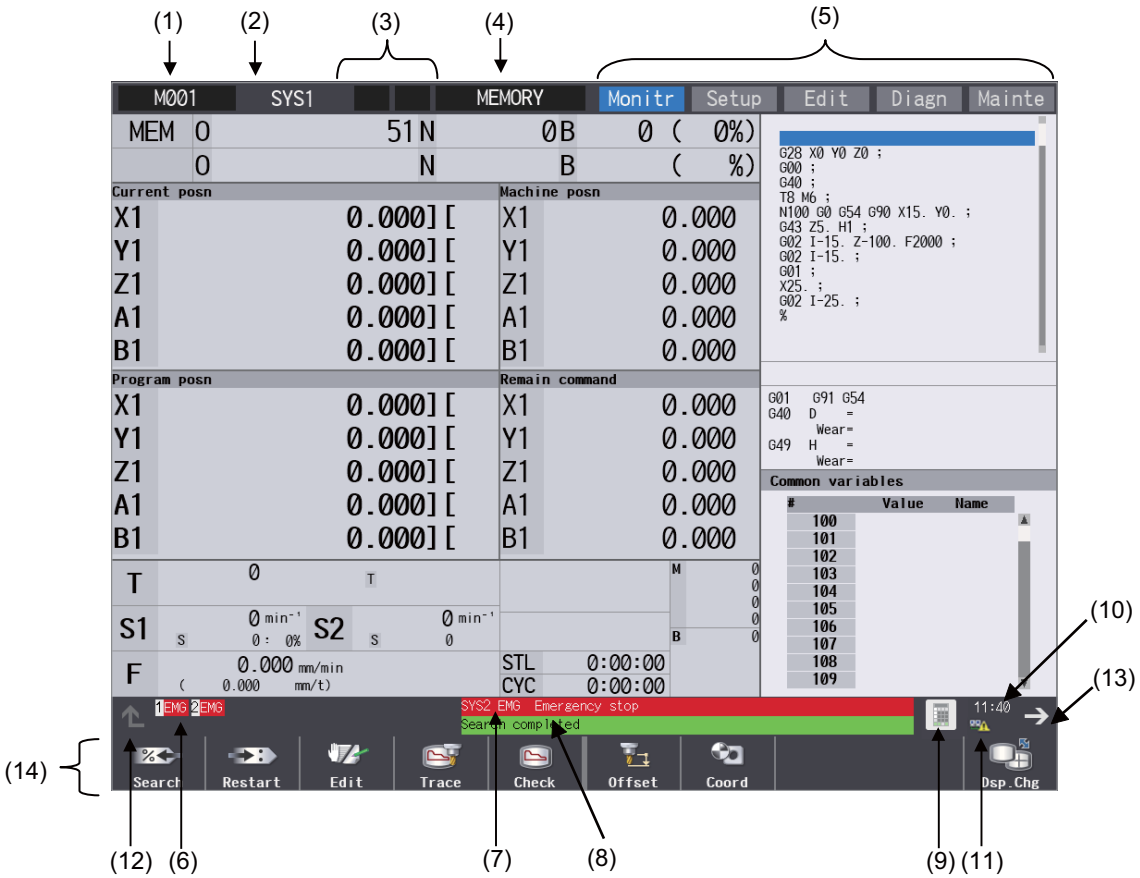
2.1.1 Screen Configuration

- 8.4-type/10.4-type





- 15-type/19-type



Display items

Display items		Details
(1)	Unit name/icon display/ transition button to home screen	The display is changed in the following order of priority: 1. Transition button to home screen When the home screen display is valid by the parameter "#11080 HomeScreen display", the transition button to home screen is displayed. When the home screen display is invalid, either icon display or unit name is displayed. 2. Icon display When there is an icon prepared by MTB, the icon is displayed. When there is no icon prepared, the unit name is displayed. 3. Unit name The currently displayed unit name (name set in the parameter "#1135 unt_nm") is displayed. When the unit name is not set, it is not displayed.

Display items		Details
(2)	Part system name	When you use the multi-part system, the name of currently displayed part system is displayed. The part system name is not displayed for 1-part system. <Main part system> The name set in the parameter "#12144 system name2" is displayed. When no value is set in #12144, the name set in the parameter "#1169 system name" is displayed. When no value is set in both #12144 and #1169, "$ + (part system No.)" is displayed. (Example) "$2" for the 2nd part system <Sub part system> The name set in the parameter "#12145 SBS_name2" is displayed. When no value is set in #12145, the name set in the parameter "#12059 SBS_name" is displayed. When no value is set in both #12059 and #12059, "SUB + (part system No.)" is displayed. (Example) "SUB2" for the 2nd part system
(3)	NC Status	The current NC status is displayed. When the status is multiple, the high-priority contents are displayed. For details, refer to "2.1.3 NC Status".
(4)	Operation mode/MDI status	The displayed part system's operation status is displayed. The MDI status is also displayed when the MDI operation mode is selected.
(5)	Screen group	The currently selected screen group is displayed.
(6)	Operation status	The NC operation status is displayed.
(7)	Alarm message	The highest-priority message among the currently occurring alarms or warnings is displayed.
(8)	Operation message	The operation message is displayed.
(9)	Software keyboard button	When the button is touched, the software keyboard is displayed. The software keyboard button is not displayed when the parameter "#11010 Software keyboard" is set to "0". However, when the NC keyboard is not mounted, the software keyboard button is displayed regardless of the parameter setting.
(10)	Time	The current time is displayed. (hour:minute)
(11)	Host connected status	The icon is displayed under the following conditions. ♦When the parameter "#8931 Display/Set limit" is set to "0" or "1", and the other host computer or display unit is connected. ♦When the parameter "#19701 Restrain VNCserver" is set to "1" or "2", and the VNC client is connected. (*1)
(12)	Menu return button	The displayed screen operation menu is switched to the screen selection menu of the current screen group. This is also used to cancel the menu operations of the displayed screen.
(13)	Menu change button	When all menus cannot be displayed at a time, the menus other than currently displayed menus are displayed.
(14)	Menus	This is used to switch the screen, select the screen specific operation, etc.
(15)	Window name	Then name of currently displayed window is displayed.
(16)	Close button	When the button is touched, the window is closed.

(*1) For details on the VNC server, refer to "17 Appendix 4: VNC Server Function".

2.1.2 Displayed Part System

Changing the part system with $\leftrightarrow$ key

Press the $\leftrightarrow$ key to change the displayed part system.

The displayed part system No. is counted up by one each time the key is pressed. If the displayed part system No. exceeds the parameter "#11055 Disp. sysno", the displayed part system No. returns to "1".

Changing the part system with a touchscreen

Displayed part system will change when the part system name on the touchscreen is pressed. The displayed part system No. is counted up by one each time the key is pressed. If the displayed part system No. exceeds the parameter "#11055 Disp. sysno", the displayed part system No. returns to "1".

Restrictions when changing the displayed part system

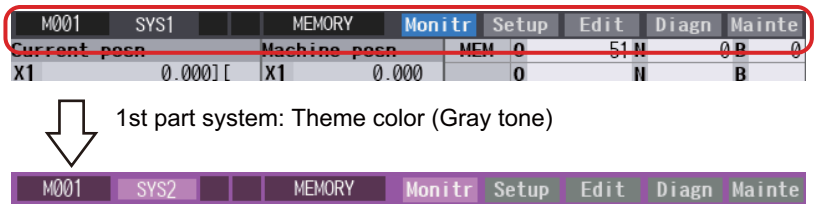
The operation to change the displayed part system by the $\leftrightarrow$ key or touchscreen is restricted by the setting of parameter "#11035 Sys. change limit".

Setting the color of displayed part system

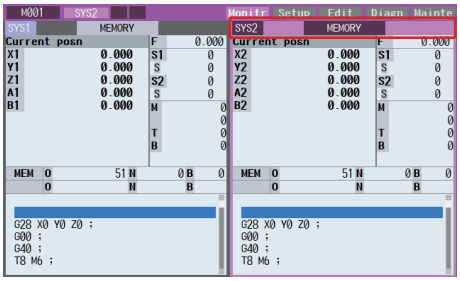
The color combination for the top left of the screen, window title, and input area can be changed with the setting of parameters "#8942 $1 color" to "#8945 $4 color" and "#8962 $5 color" to "#8965 $8 color". Each part system can have a unique color combination which makes easy to understand the displayed part system.
When the part system is changed, the upper left of the screen will change colors according to the settings for each parameter.

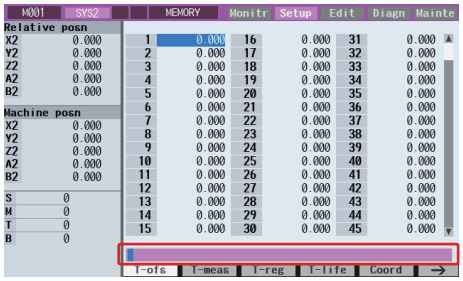
[Setting example]

When parameter "#8942 $1 color" is "1", parameter "#8943 $2 color" is "2", and parameter "#11060 Screen theme color" is "1".



When the parameter "#11019 2-system display" (2-part system simultaneous display) is "1" or "2", the part system names, operation mode, and MDI status are displayed in the title bar, and the active part system is displayed in the color specified in the parameter for the part system. Also, the background color in the input area will be matched to the part system color.

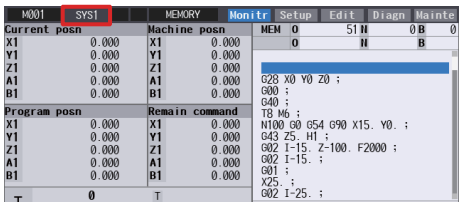




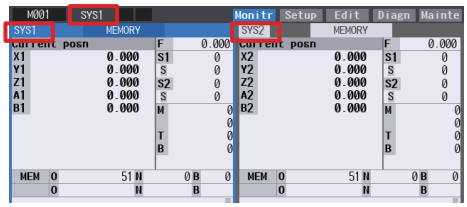
Displaying the sub part system name

When the part system set in the parameter "#12049 SBS_no" (Sub part system identification No.) operates as sub part system, sub part system name set in the parameter "#12145 SBS_name2" (Sub part system name 2) can be displayed as part system name.
Display area for the part system names shown in the following figures can switch main/sub part system display according to NC operation status.

[1-part system display]



[2-part system display]



2.1.3 NC Status

The status display area at the top left of the screen indicates the current NC operation status. Nothing is displayed when status is normal.
When the NC operates with three or more status, the higher-priority two status are displayed. The items in the next table are listed in order of priority.
While any status is being displayed, this indicator flashes at intervals of approximately 1 second. Wireless status is displayed without flashing.

Display position	Status	Definition
Left side	PR	The parameter has changed, and it requires turning the power ON again. The parameter will be validated after turning the power ON.
		The status displayed when the wireless LAN function is enabled. Refer to the following table for more details.
Right side	TM	Indicates that measurement is performed by on-board servo tuning.
	TR	Indicates that measurement by on-board servo tuning is ready. The measurement is started by pressing the cycle start button.
	AT	Indicates when auto tuning is valid. In this case, the automatic start is disabled.
	HC	Indicates when the high cycle sampling is valid.
	RN	Indicates when selecting machining surface.
	BT	Indicates when backlash auto adjustment is ON.
	EM	Indicates when performing workpiece installation error measurement or rotation center error measurement.
	HA	Indicates when adjusting high-accuracy parameters and operates using the adjusted high-accuracy parameters.

Operation mode	Status	Definition
Station (STA) mode		Indicates that the NC is not connected to the connection access point (AP).
	 (Strong)	Indicates that the NC is connected to the connection access point (AP). The displayed icon will change according to the signal strength.
	 (Medium)	
	(Weak)	
Access point (AP) mode		Indicates that the NC is not connected to any stations (STAs).
		Indicates that the NC is connected to one or more stations (STAs).

For details on wireless functions, refer to "15 Appendix 2: Wireless LAN Function [M800VS/M80V Series]".

2.1.4 Operation Mode/MDI Status

The displayed part system operation mode and MDI status are displayed.
When the operation mode or MDI status is changed, the display changes as follows.

Display	Details	Details
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 NON	No MDI setting	Automatic operation is performed with the program set in the MDI screen.
MDI SET	MDI setting completed	
MDI RUN	MDI running	
JOG	Jog	This mode enables an axis to be moved by hand consecutively at the feed rate set by the feed rate selection switch.
HANDLE	Manual handle	This mode enables an axis to be moved by turning the manual handle. The travel distance per graduation of the handle depends on the value set by the handle magnification switch.
STEP	Step	This mode enables an axis to be moved by manual feed rate when the feed axis selection switch is ON. The travel distance per graduation of the handle depends on the value set by the incremental magnification switch.
MANUAL	Manual arbitrary feed	Moving by an arbitrary amount and the arbitrary positioning can be performed in this mode.
ZERO-RTN	Reference position return	This mode enables a control 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 an axis to manually be pushed against the machine end stopper and then performs the zero point return operation.
RAPID	Rapid traverse	The rapid traverse feed mode enables the machine to be moved consecutively at rapid traverse rate manually.
JOG+HND	Jog + Handle	This is the jog and handle synchronous feed mode.
RAP+HND	Rapid traverse + Handle	The rapid traverse and handle can be used simultaneously.
NO MODE	No operation mode	The operation mode is not selected.

2.1.5 Operation Status

The operation status displays the currently selected NC operation status for each part system. (The maximum number of the part system which can be displayed.)
The inverted number expressing the part system No. is not displayed for 1-part system.

Symbol	Details	Character color	Background color
EMG	In emergency stop	White	Red
RST	Resetting NC	White	Dark gray
BST	In block stop	White	Dark gray
HLD	Operation halted	White	Dark gray
SYN	Synchronizing	White	Dark gray
CRS	Cross conversion waiting	White	Dark gray
FIN	Miscellaneous function (M, S, T, 2nd miscellaneous function) completion waiting (*1)	White	Dark gray
AUT	In automatic operation	White	Dark gray
RDY	Operation completed state	Black	Green

(*1) This symbol is displayed when the parameter "#1725 cfg25/bit1" is set to "1".

2.1.6 Alarms/Warnings

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

(Example) Warning message display

SYS2 M01 No operation mode

Type	Details	Character color	Background color
NC alarm message	The operation alarm, program error, MCP alarm, servo alarm, or system alarm which is currently occurred is displayed.	White	Red
NC warning message		Black	Yellow
PLC alarm message	The message such as the machine error contents is displayed using user PLC.	White	Red
PLC operator message	The operator information message is displayed using user PLC.	Black	Yellow
NC alarm message at the background check	The message is displayed when a program error occurs during the check, or when the macro alarm message (message display by #3000 variable command) is displayed.	White	Orange

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

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)

2.1.7 Operation Messages

(Example)

Search completed

Type	Details	Character color	Background color
Operation messages	This displays the messages for operation.	Black	Yellowish green

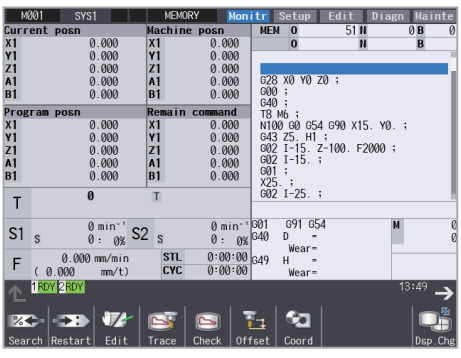
The operation message can be reset by pressing any key.

2.1.8 Changing the Screen Theme Color

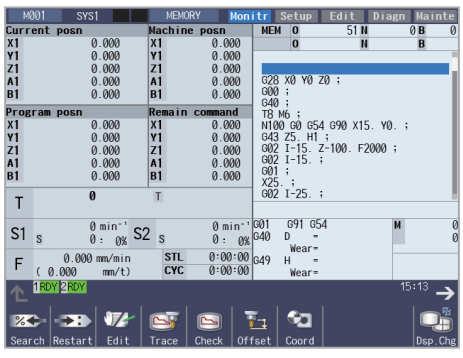
The theme color (display color) of the entire screen can be changed with the setting of the parameter "#11060 Screen theme color".

Monitor screen examples are shown below.

[When parameter #11060 is "0" (gray tone)]



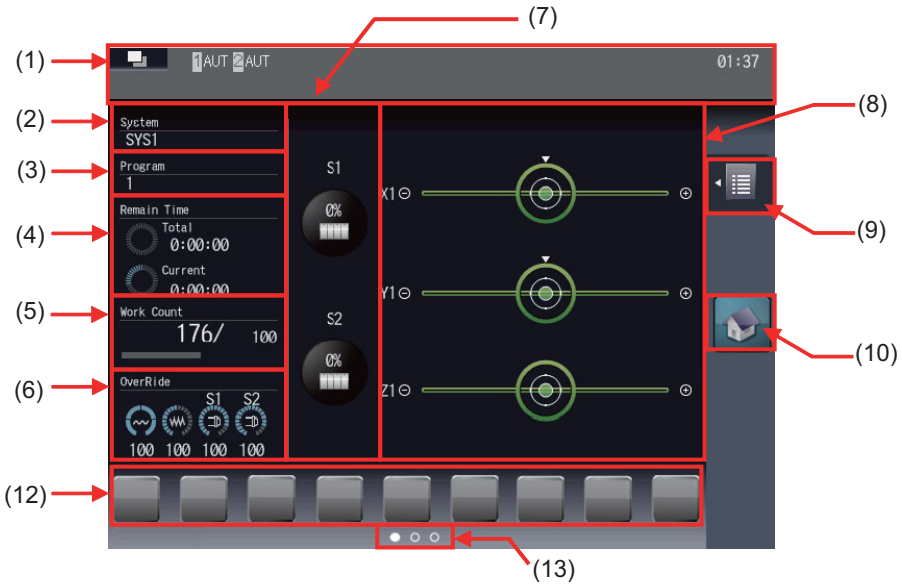
[When parameter #11060 is "1" (blue tone)]



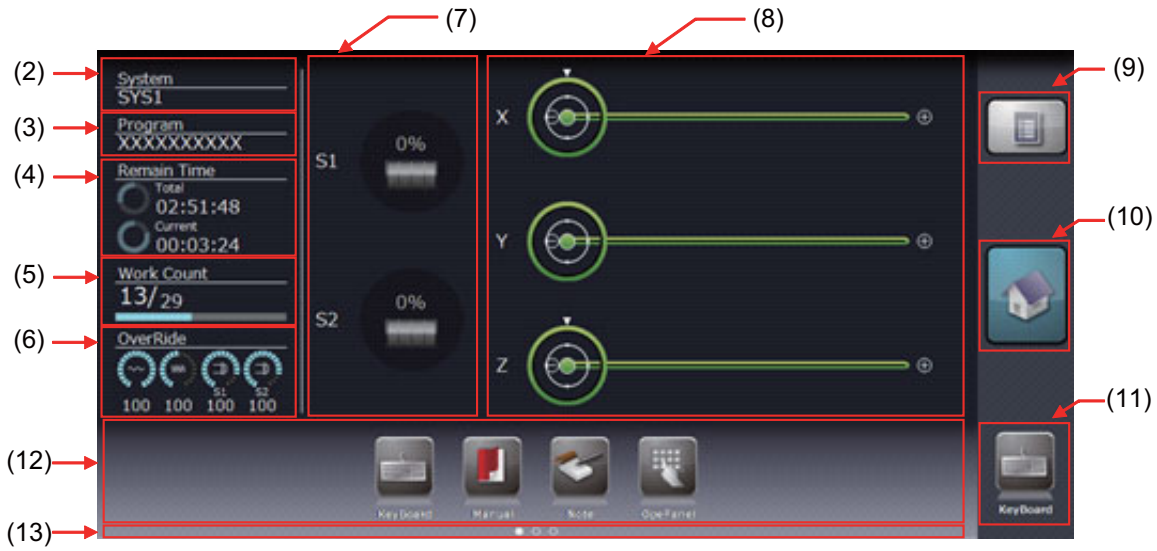
2.2 Home Screen

Refer to "1.3 Standard Screen and Home Screen" for general outline of the home screen.

2.2.1 Screen Configuration

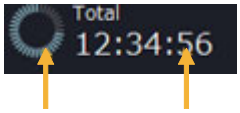


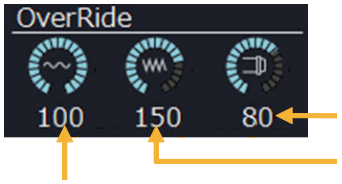
<For 19-type vertical display unit>



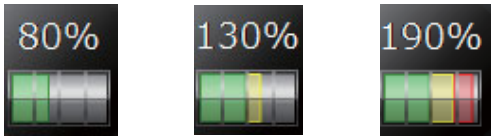
Display items

Display items	Details
(1) Fixed display part (*1)	The following items are displayed. •Standard screen switch button When you touch the button, the screen is changed to the previously displayed standard screen. The window display key ( ) performs the same function. •Operation status The NC operation status is displayed. •Alarm messages Currently occurring alarm or warning is displayed. For details, refer to "2.1.6 Alarms/Warnings". •Operation messages •Time (h:m)
(2) Part system name	When you use the multi-part system, the name of currently displayed part system (name set in the parameter "#1169 system name", or name set in the parameter "#12059 SBS_name" for operating as the sub part system) is displayed. The part system name is not displayed for the 1-part system. When the values are not set in the parameter, "$ + (part system number)" is displayed. (Example) "$2" for the 2nd part system When the values are not set in the parameter for the name of sub part system, "SUB + (part system number)" is displayed. (Example) "SUB2" for the 2nd part system <For 19-type vertical display unit> The selected part system's name is displayed as follows: The name set in the parameter "#12144 system name2" is displayed. When no value is set in #12144, the name set in the parameter "#1169 system name" is displayed. When no value is set in both #12144 and #1169, "SYS + (part system No.)" is displayed. (Example) "SYS2" for the 2nd part system
(3) Machining program	Displays the name of machining program being in the operation search. The name of main program is displayed when the main program is being executed, and the name of sub program is displayed when the sub program is being executed. If the program name is 14 characters or longer, first to 13th characters are displayed.

<table>
<tr><th colspan="2">Display items</th><th colspan="2">Details</th></tr>
<tr><td>(4)</td><td>Remaining machining time</td><td colspan="2">Displays the remaining machining time and reference information of progress rate based on the time from the cycle start to the M command specified with the parameter "#8001 WRK COUNT M" or from the cycle start to M02/M30 command, that is memorized as the machining time per one machining.
To display the total time, the workpiece machining count (parameters "#8001" to "#8003") must be specified.
<Note>
♦Time of one machining program which is most recently executed is memorized as the machining time.
♦If the target program has never been executed after the operation search, the remaining machining time and remaining machining rate are not displayed, but "--:--:--" is displayed.
♦When the operation search or restart search is executed, the remaining machining time and remaining machining rate are not displayed, but "--:--:--" is displayed.
♦When the restart search or sequence No. search is executed, the time from the search point is memorized as the machining time.
♦When reset is performed during the machining, the display returns to the previous state before the machining. When re-machining is performed after the reset, the time from the beginning of re-machining is memorized as the machining time.
♦When the integrated time is set with variables during the machining, the memorized machining time may not be displayed properly.</td></tr>
<tr><td></td><td></td><td>Total</td><td>Displays the remaining time and progress rate of the machining for all workpieces.
<Display example>
Progress rate of the machining = 25% and the remaining machining time = 12 hours 34 minutes 56 seconds


Progress rate of machining
Remaining machining time</td></tr>
<tr><td></td><td></td><td>Current</td><td>Displays the remaining time and progress rate of the machining of a workpiece.</td></tr>
<tr><td>(5)</td><td>Work count</td><td colspan="2">Displays the number of machining times at present (number of machined workpieces/total number of workpieces to be machined) and the proportion.
The display depends on the number of workpiece machining that has been set with the parameters "#8001 to #8003".
However, when the parameter "#8001 WRK COUNT M" is set to "0", M command is not counted and the work count is not displayed.</td></tr>
<tr><td>(6)</td><td>Override</td><td colspan="2">Displays the rapid traverse override (%), cutting feed override (%) and spindle override (%).
The rapid traverse override is displayed in the range of 0 to 100%, the spindle override is displayed in the range of 0 to 120%, and the cutting feed override is displayed in the range of 0 to 200%.
<Display example>
Rapid traverse override 100%, cutting feed override 150%, and spindle override 80%


Spindle
Cutting feed
Rapid traverse</td></tr>
</table>

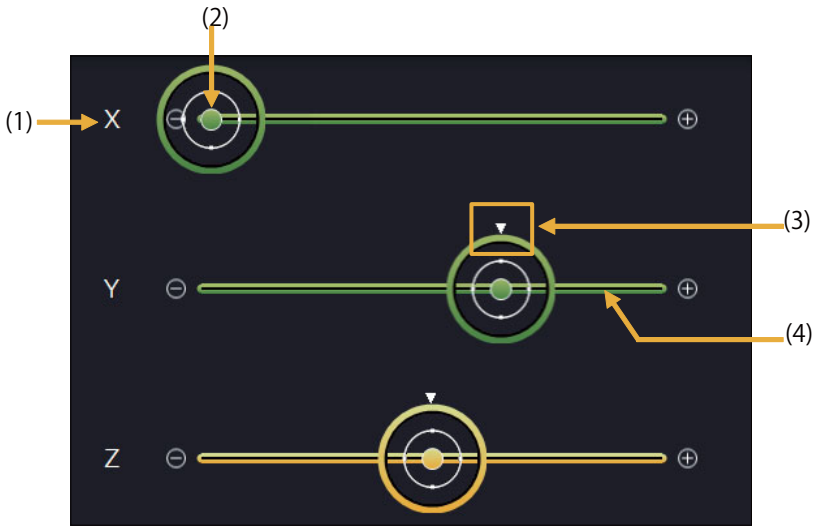
Display items	Details
(7) Spindle	Displays the spindle load rate (%) with the graph and numeric. Up to two spindles can be displayed per part system. The displayed color of graph varies depending on the spindle load condition. The thresholds of the displayed color (caution/warning) are made in accordance with the parameters (#3192 to #3194). <Display example> (Normal: Green) (Caution: Yellow) (Warning: Red) 
(8) Machining axis	Displays the position and load condition (load) of linear axes and the position of rotary axes by animation. Up to three linear or rotary axes can be displayed per part system. Refer to the explanation in next section for details.
(9) Operation menu button	Displays the operation menu for each screen. Home screen: Operation menu to switch the displayed part system Each application: Operation menu for each application (Refer to the specifications issued by MTB for details.) Touch the operation menu button to display/perform the menu. The same operation can be performed with the following keys: Show/hide the operation menu: menu list key (LIST) Select/move the menu: cursor keys Determine the menu: [INPUT] key
(10) Home button	Returns to the home screen.
(11) Shortcut button (*2)	This is the application button to switch to the application set by MTB and always displayed on any screen. Two applications can be registered on the shortcut button, and first registered application is usually displayed on the shortcut button. The second registered application is displayed on the shortcut button during the activation of the first registered application.
(12) Application button	Switches to each registered application. Application buttons can be deleted.
(13) Page position	Displays the page position of application buttons.

(*1) This is not displayed on the 19-type vertical display unit.

(*2) This is displayed only on the 19-type vertical display unit.

Machining axes display

The position and load condition of linear axes and the position of rotary axes are displayed by animation.



Display items		Details
(1)	Name of linear axis	Displays the specified linear axis name which has been set with "#1022 axname2". Only linear axis name is displayed, but the rotary axis name is not displayed.
(2)	Linear axis position display	Indicates the linear axis position (disk-shaped figure) by animation. When the axis moves, the disk-shaped figure shifts right and left in accordance with the machine position. The range of coordinate position is determined by the parameters "#2013 OT-" and "#2014 OT+". If the parameters are not set, the operation is performed in the range of -1000.000 to 1000.000. <Display example> OT- = -40.000, OT+ = 300.000, and Machine position = 130.000 OT- (-40.000) OT+ (300.000)  Machine position (130.000)
(3)	Rotary axis position display	Indicates the rotary axis position (triangle mark on the disk-shaped figure) by animation. The triangle mark moves to upper side of the disk-shaped figure for 0 degree and moves to lower side of the disk-shaped figure for 180 degree.
(4)	Linear axis load condition display	The displayed color of the bar changes depending on the linear axis load condition (load value). The bar is colored with green for normal state, yellow for caution and red for warning. The thresholds of the displayed color (caution/warning) are made in accordance with the parameters "#2643 LdMeter thresholdY" and "#2644 LdMeter thresholdR".

2.2.2 Switching Displayed Part System from the Operation Menu

When you touch the operation menu [System change] and select the part system (SYS1, etc.) that appears at the touch, the display switches to the selected part system.

(Example: Switch to 2nd part system)

(1) Touch the operation menu button. → The operation menu is displayed.



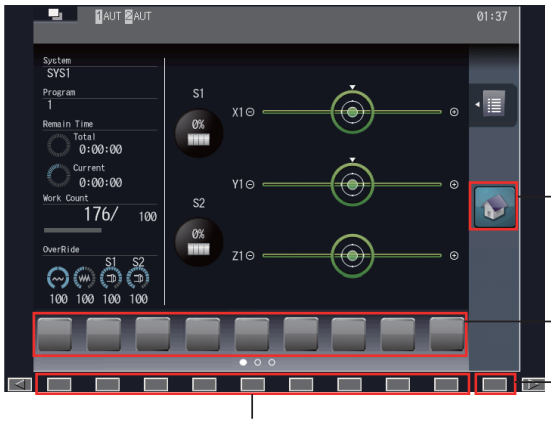


(2) Touch the menu [System change]. → Names of selectable part systems are displayed. Currently selected part system is highlighted.

(3) Touch [SYS2]. → The operation menu is closed, and the displayed part system is changed to the 2nd part system.

2.2.3 Operating the Application

To move to the application screen, touch the application button on the bottom of the screen.
The same operation can be performed by touching the 1st to the 9th menu keys from the left.



Touch the home button to go back to the home screen.
The same operation can be performed by touching the 10th menu key from the left.
For a full-screen application, however, follow the operation specifications of each application.

Operating the application menus

The operation for each application can be selected by touching the operation menu button () or pressing the menu list key (

).
However, as some applications are displayed in full-screen mode, the operation menu button may not be displayed.

■Operating with the operation menu button

(1) Touch the  button. → The main menu is displayed.

(2) Select the main menu. → The sub-menu is displayed on the left of the main menu.

(3) Touch the sub-menu to be executed. → The menu disappears and the display goes back to the application screen. On the application screen, the selected sub-menu is executed.

■Operating with the menu list keys

(1) Press the 

key. → The main menu is displayed.

(2) Use the cursor key [↑] or [↓] to select the main menu.

(3) Press the cursor key [←]. → The sub-menu is displayed on the left of the main menu.

(4) Use the cursor key [↑] or [↓] to select the sub-menu to be executed.

(5) Press the [INPUT] key. → The menu disappears and the display goes back to the application screen. On the application screen, the selected sub-menu is executed.

Note

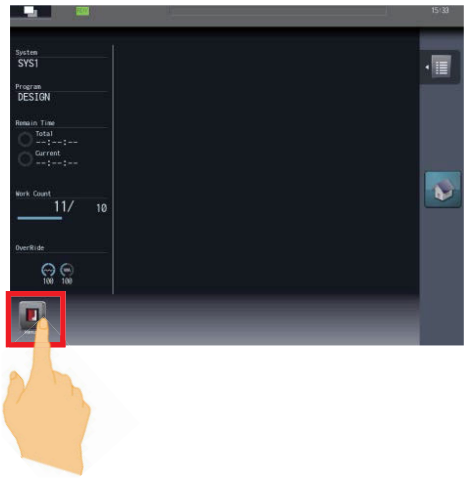
(1) If you touch any other part than the operation menus while they are being displayed, the menus are closed.

2.2.4 Displaying the Manuals [M800VW Windows-based Display Unit/M80VW Windows-based Display Unit]

To display each manual on the screen, press the manual button assigned to an application button.
Application buttons are assigned to 1st to 9th menu keys from the left. The same operation can be performed by pressing the menu key which the manual button is assigned to.
The displayed manuals differ depending on MTB.

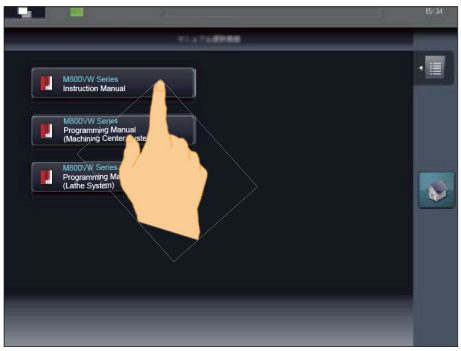
Displaying the selected manual

(1) Touch the manual button.



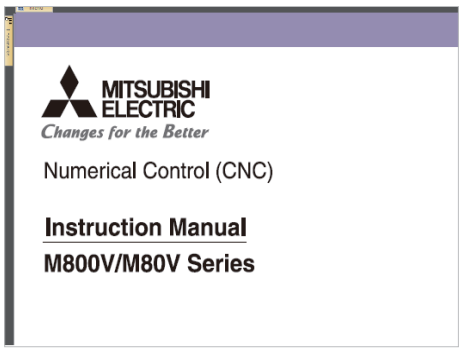
→ The manual selection screen is displayed.
Manuals which can be displayed are shown in the list.
The cursor is displayed at the manual button on the top left.

(2) Touch the manual to be displayed.



When you operate with keys, move the cursor with [↑], [↓], [←] or [→] key and determine with the [INPUT] key to display the selected manual.

→ The selected manual is displayed.



Note

(1) When you move to another screen by the function keys, window display key ( ) or direct screen selection while the manual is displayed, the designated screen is displayed after the displayed manual is closed.

(2) When you press the window display key ( ) after the display is moved from the manual selection screen or manual display screen to the standard screen, the home screen is displayed.

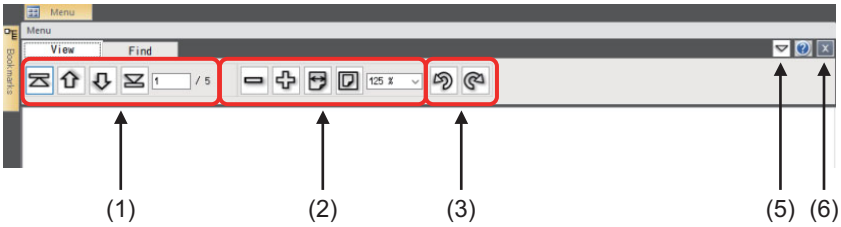
Operation button on the manual screen

■Normal display

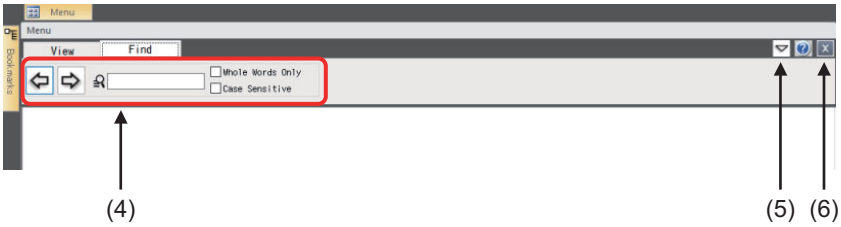


Touch the [Menu] tab to switch to the ribbon display. The ribbon displays the operation buttons such as page switching, enlarging/reducing, rotation and search.
Touch the [Bookmark] tab to display the contents of the manual.

■Ribbon display (View tab)



■Ribbon display (Find tab)



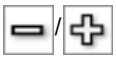
■Bookmark display



	Display items	Details
(1)	Page jump group	This displays the operation buttons related to the displayed page or page jump. : This displays the first page. : This displays the previous page. : This displays the next page. : This displays the last page. : Page number input area This displays "the current page number/the total number of pages" of the manual.
(2)	Display magnification group	This displays the operation buttons related to the display magnification. : This reduces the display. : This enlarges the display. : This displays the manual based on the page width. : This displays the whole page of the manual. : This displays the magnification. You can select the display magnification from the drop-down list.
(3)	Rotation group	This displays the operation buttons related to the rotation. : This rotates the page 90 degrees to the left. : This rotates the page 90 degrees to the right.
(4)	Search group	This displays the operation buttons related to the search group. : Search character(s) input area : This searches for the previous item. : This searches for the next item. Search option You can search with the following conditions by checking the box: Whole Words Only: This searches for an exact-match of the given string. Case Sensitive: This searches for the given string distinguishing upper- and lower-case letters.
(5)	Ribbon minimize/expand button	This minimizes the ribbon display to return to the normal display.
(6)	Close button	This closes the manual display screen to return to the manual selection screen. You can also close the manual by pressing the menu changeover key ().
(7)	Bookmark	This displays the bookmarks of the manual being displayed. Select an item from the contents to move to the corresponding page.

Enlarging/reducing the display

To enlarge/reduce the display, use the following methods:

- Pinch in to reduce and pinch out to enlarge on the manual. (*1)
- Touch the [Menu] tab to operate the buttons of the magnification display group in the ribbon display.
  - Touch  button.
  - Select the magnification from the drop-down list of the magnification display.
- Press the [+]/[-] key.

(*1) Pinch in: A gesture to pinch a target between the thumb and index finger on the screen

Pinch out: A gesture to spread the thumb and index finger on the screen

Displaying the whole page

When you touch the [Menu] tab and then  in the ribbon display, the height of a single page of the manual is displayed with the size which fits the window.

Touch  to undo.

Moving between pages

To move between pages, use the following methods:

■Moving to the next page

- Drag or flick upwards on the manual. (*1)
- Drag the vertical scroll bar on the right of the screen downwards.
- Press the cursor key [↓] or the next page key ().

■Moving to the previous page

- Drag or flick downwards on the manual. (*1)
- Drag the vertical scroll bar on the right of the screen upwards.
- Press the cursor key [↑] or the previous page key ().

■Moving to the designated page

Touch the [Menu] tab, input the intended page number in the page number input area in the ribbon display and then press the [INPUT] key.

(*1) Drag: A gesture to touch the surface and move your finger up and down without losing contact

Flick: A gesture to quickly run your fingertip across the screen

Rotating the display

Touch the [Menu] tab to rotate the page using  in the ribbon display.

Searching for a word

(1) Touch the [Menu] tab and then [Find] in the ribbon display.

(2) Input a word to be searched for in the search character(s) input area and press the [INPUT] key. → The page containing the search word appears. The corresponding area is highlighted.

(3) Touch the  button. → The cursor moves to the next corresponding area. If there is no corresponding area afterwards, search completion message appears.

Note

(1) If you touch any area other than the search character(s) input area, you cannot input character(s). To input the search character(s) again, touch the search character(s) input area.

(2) Only alphanumeric characters and symbols can be input for the search character(s).

(3) When the search reaches the end of the manual, the search is performed returning to the top of the manual. When the search reaches the search start point, search completion message appears.

2.3 Home Applications (19-type Vertical Display Unit)

Refer to "1.3 Standard Screen and Home Screen" for general outline of the 19-type vertical display unit.

2.3.1 Home Screen (Machine Status Display)

Refer to "2.2.1 Screen Configuration" for details of the display items on the home screen.

Switching to a registered application

The screen can be switched to the registered application by touching the shortcut button or one of application buttons.

Note

(1) The expansion application registered to the shortcut button differs depending on MTB specifications.

Switching to the home screen

Each application screen can be switched to the home screen (machine state display) by touching the home button.

Switching the pages of application buttons

The pages can be switched by flicking the application button area.

Deleting an application button

Each application button can be deleted by touching the delete menu.

(1) Touch and hold an application button to be deleted for seconds.

(2) Release your finger when the application button icon becomes the movable state.

(3) The delete menu appears. Touch the delete menu.

(4) Touch the [OK] button in the pop-up window to delete the application button.

Rearrange the application buttons

Application buttons can be rearranged by drag-and-drop operation.

Note

(1) When the application buttons on the page you moved are maximum numbers (7 pcs), the latest registered icon goes to next page.

Changing the displayed part system for the home screen

A displayed part system can be switched by selecting the name of part system to be displayed (SYS1, etc.) on the menu [System change] of the operation button.

(1) Touch the operation menu button.

(2) Touch [System change] on the main menu. Selectable part system names are displayed on the sub menu. Select a part system name to be displayed, and the display changes.

Registering an application button

Application buttons can be registered from the application list displayed on the menu [Add Application] of the operation menu button.

(1) Touch the operation menu button.

(2) Touch [Add Application] on the main menu.

(3) The list of applications which can be registered to application button is displayed on the sub menu.

(4) Touch the application to be registered.

(5) Touch the [OK] button in the pop-up window to add the application button.

Note

(1) When the displayed page contains the maximum number of application buttons (seven buttons), the button will be added to the end of next or succeeding page which contains the application buttons less than seven. The page of which the application button is added is displayed.

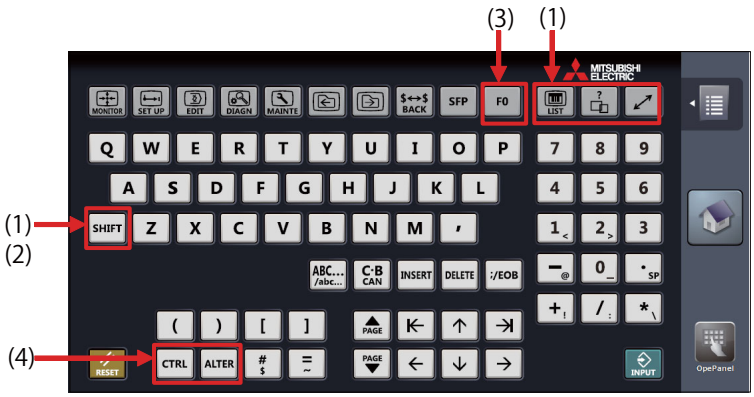
(2) If all pages contain the maximum number of application buttons (seven buttons), the error message appears.

Displaying the application version

The version of application for 19-type vertical display is displayed by [Version] of the operation button.

2.3.2 Keyboard Application

The keyboard application enables the key input by touching buttons.



For details on the keys, refer to "1.1 Display Unit and Keyboard Unit".
This section explains only keys which have different meanings with the keyboard application.

Key type		Key	Operation
(1)	Window operation key	LIST	Displays the menu list window.
		?	Displays the parameter/operation guidance window.
		SHIFT + LIST	Switches the prototype screen and mass production screen.
		SHIFT + ?	This varies depending on the MTB specifications.
		↗	Switches the active window.
(2)	Shift key	SHIFT	Enables the lower column of each key. Once the [SHIFT] key is selected, the [SHIFT] key remains in effect until any other keys are touched or the home screen is restored. The menu is highlighted while the [SHIFT] key is enabled.
(3)	F0 key	F0	Displays the on-board screen or MTB created screen. <Note> ♦This action differs depending on MTB specifications
(4)	Modifier key	CTRL ALTER	Once the [CTRL] (or [ALTER]) key is selected, the key remains in effect until any other keys are touched. When the home screen is restored, the [CTRL] (or [ALTER]) key is also canceled.

Note

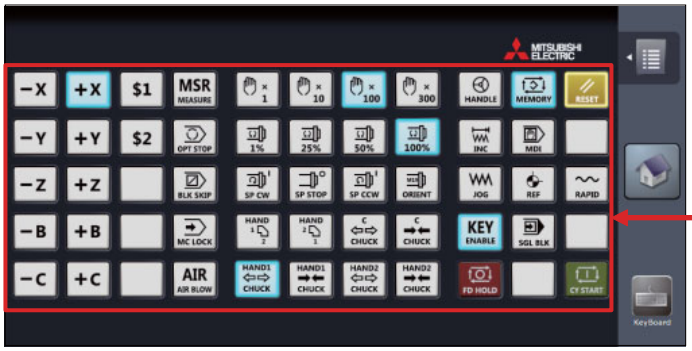
(1) For eight types of keys; ↑, ↓, ←, →, PAGE↑, PAGE↓, |←, and →| the key is continuously input (key repeat state) during being touched.

(2) The ABC.../abc... key is interlocked with the CapsLock key of Windows. By pressing the ABC.../abc... key, the input characters on the operation panel are also changed uppercase and lowercase.

(3) The key is input when it is touched. Even when sliding a finger and releasing another key, only first key is input.

2.3.3 Operation Panel Application

The operation panel application enables the same operation as the switches and buttons of the actual machine operation panel, such as operation mode selection, automatic operation start and override setting.
Refer to the instruction manual issued by the MTB for meanings and layout of buttons.



Pressing each machine operation button turns ON the PLC input signal which has been assigned to the button. When the PLC output signal which has been assigned to the button becomes ON state, the button color turns to that indicates ON state.

Operating the machine on operation panel

(1) When the machine operation button is touched, the button is changed to the pressed state color, and the X device signal which has been assigned to the button is turned ON. The device signal input is performed the moment the button is touched.

(2) The button returns to the normal color when the operation panel button is released.

Note

(1) When the PLC output signal turns ON, the button turns the ON state color (blue in the example above).

2.3.4 Manual Application

The manual application enables to refer to each manual. Text search, page jump with touch operation, zooming in/out display, etc., are available.

There are two display methods for the manual application. This section explains the operation using the display method 1 of the manual application. For the operation using the display method 2, refer to "2.2.4 Displaying the Manuals [M800VW Windows-based Display Unit/M80VW Windows-based Display Unit]".

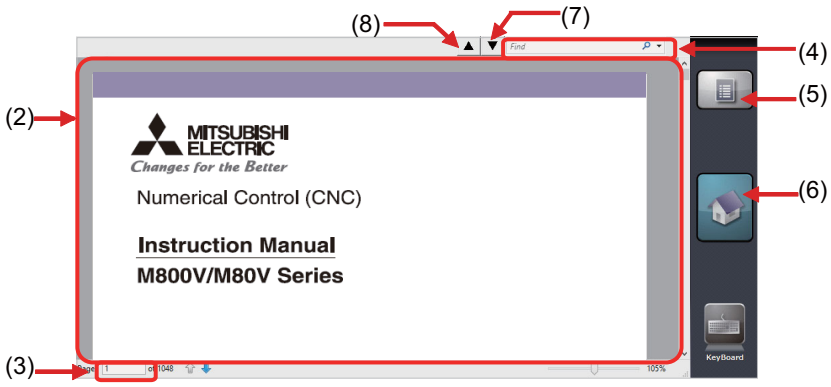
Note

(1) The displayed manuals and methods differ depending on the MTB.

Screen image example (file selection screen/manual display screen)



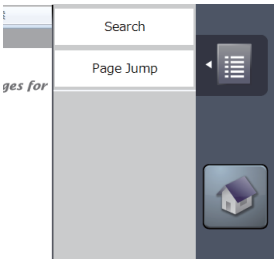
■Display method 1



■Display method 2



Display items

	Display items	Details
(1)	Manual button	Opens the manual when it is touched.
(2)	Main window	Displays the contents of the manual.
(3)	Page No. input area	Input a page number for page designation.
(4)	Retrieval input area	Input a search string for retrieval.
(5)	Operation button	The following menus are displayed by pressing this button. Search: This performs the search. Page jump: This moves to the designated page. 
(6)	Home button	Returns to the home screen.
(7)	Find next	Displays the next retrieval point.
(8)	Find previous	Displays the previous retrieval point.
(9)	[Menu] tab	This displays the operation buttons such as page switching, enlarging/reducing, rotation and search.
(10)	[Bookmark] tab	This displays the bookmarks of the manual.

Selecting a manual

(1) Activate the manual application, and the file selection screen appears.

(2) Flick on the file selection screen to change the page.

(3) Select (touch) a manual to be referred, and the selected manual is displayed.

Scrolling the manual

Flick the main window. When sliding your finger to up, the page moves to up and next page is displayed.

Searching the characters in the manual

(1) Touch the operation button on the right of screen.

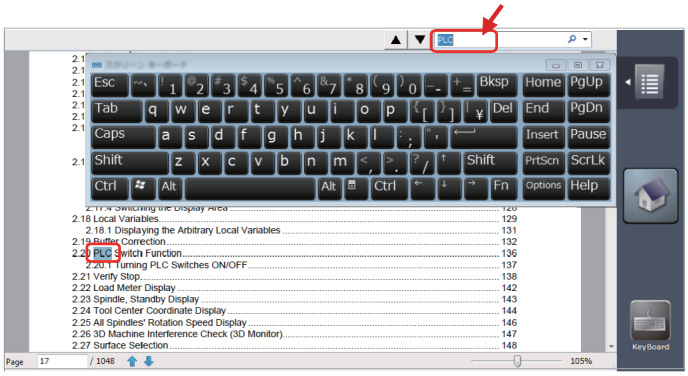
(2) The sub menu appears. Touch [Search].

(3) The cursor appears in the retrieval input area and the screen key board starts.

(4) Input a search string with the screen keyboard and press the Enter (⏎) key. The point at which the search string is found will be displayed.
Press the Enter key again, and next point will be searched.

Touch the close button (☒) in the top right to close the screen keyboard when the screen keyboard becomes unnecessary.

Search character(s)



(5) To continue the retrieval, press the [▼] button to find the next point and press the [▲] button to find the previous point.

Note

(1) The Caps key of the screen keyboard also switches the upper case/lower case input on the standard screen.

(2) When the area other than the retrieval input area is touched, the search string cannot be input. To input the search string again, touch the retrieval input area.

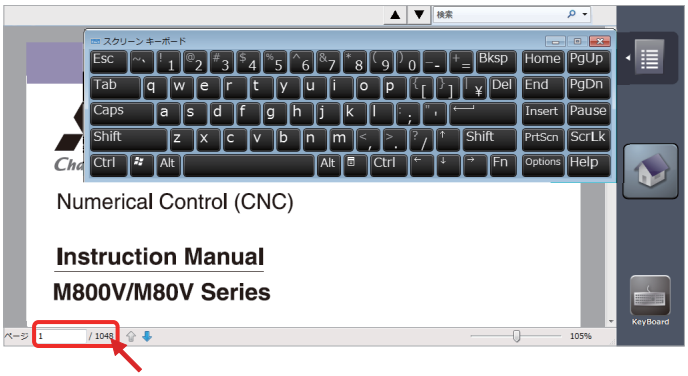
(3) Only alphanumeric characters and symbols can be input.

Displaying the manual by page designation

(1) Touch the operation button on the right of screen.

(2) The sub menu appears. Touch [Page Jump].

(3) The cursor appears in the page number input area and the screen key board starts.



Page No. input area

(4) Input a page number to be referred and press the Enter key. The designated page is displayed.
Touch the close button in the top right to close the screen keyboard when the screen keyboard becomes unnecessary.

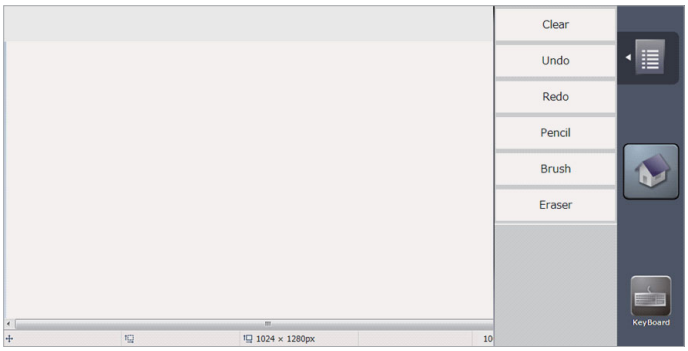
Note

(1) The front page of the manual file is the first page for the page number designation.

2.3.5 Note Application

The note application enables the hand writing with the touch operation. This can be used as a memo.
The written contents are saved automatically when the home screen is restored, and the contents will be displayed next time the note application is started.

Screen image (when the operation button is pressed)



Display items

Display items	Details
Memo area	Characters, etc., can be written by tracing a finger.
Operation button	Performs the all erasure of memo area, undo, and various tools selection.
Clear	Erases all contents in the memo area.
Undo	Cancels a last action.
Redo	Redoes a canceled action.
Pencil	To select the pencil tool for thin lines.
Brush	To select the brush tool for thick lines. This tool is selected when the application is started.
Eraser	To select the eraser tool to erase a part of memo.

Writing a memo

Lines can be drawn with the selected tool by tracing your finger in the memo area.

When the note application is started, the brush tool which is used for drawing thick lines is selected.

Delete all contents of memo

(1) Touch the operation button on the right of screen.
(2) Touch [Clear] on the sub menu.
(3) The memo area is all deleted and the sub menu is closed.

Undo a last action

Touch [Undo] on the sub menu to cancel a last action.

(1) Touch the operation button on the right of screen.

(2) The sub menu appears. Select [Undo] on the sub menu.

(3) One action will be undone.

Note

(1) When no action is performed after the application is started from the home screen, the [Undo] action cannot be performed because there is no action can be canceled.

(2) Once the home screen is restored, the previous action cannot be canceled.

Available undo actions are as follows.

- Action between drawing a line and releasing a finger
- Action between erasing a line and releasing a finger
- [Clear] action
- [Redo] action

(3) Use [Redo] to cancel the [Undo] action.

(4) Up to 50 actions can be undone.

Redo a canceled action

Touch [Redo] on the sub menu, and the canceled action can be redone.

(1) Touch the operation button on the right of screen.

(2) The sub menu appears. Select [Redo] on the sub menu.

(3) One action taken back by the [Undo] action will be redone.

Note

(1) When no action is performed after the application is started from the home screen, the [Redo] action cannot be performed because there is no undo action performed.

(2) The [Undo] action can be redone by touching [Redo], however, once the home screen is restored, the [Undo] action cannot be redone.

Selecting the line width

(1) Touch the operation button on the right of screen.

(2) The sub menu appears. Touch [Pencil] or [Brush] on the sub menu.

(3) Touch [Pencil] to select the pencil tool to draw the thin lines, or touch [Brush] to select the brush tool to draw the thick lines.

Note

(1) The tool selection is not held after the home screen is restored. The brush tool is selected with the note application start-up.

Erasing a part of memo

(1) Touch the operation button on the right of screen.

(2) The sub menu appears. Touch [Eraser] on the sub menu.

(3) The eraser tool is selected. Only the part traced with finger can be erased.

(4) Select the pencil tool or brush tool to draw the lines again.

Note

(1) The tool selection is not held after the home screen is restored. The brush tool is selected with the note application start-up.

2.4 Setting Data

2.4.1 Setting Numerals and Alphabetical 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.
(2) Special settings may be required depending on the data type. Refer to each item.
(3) The cursor may move to the right of the display item depending on the data type.
(4) If an illegal key is set, an error occurs when [INPUT] is pressed. Reset the correct data.
(5) The behavior of the cursor after inputting data differs depending on the setting of the parameter "#1759 cfgPR09/bit1".
0: The cursor moves to the next data entry position after inputting data.
1: The cursor does not move from the original position even though the [INPUT] key or menu key operation is performed.

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.

[INSERT] key: Enters the insert mode.

The mode returns to the overwrite mode 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.

[C.B] key: Deletes all characters 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.

2.4.2 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		
Operation	**Setting example**	**Operation results**
Addition	=100+50	150
Subtraction	=100-50	50
Multiplication	=12.3*4	49.2
Division	=100/3	33.3333333
Function	=1.2 * (2.5 + SQRT(4))	5.4

Function symbols, setting examples and results			
Function	**Function symbol**	**Setting example**	**Operation results**
Absolute value	ABS	=ABS(50 - 60)	10
Square root	SQRT	=SQRT(3)	1.7320508
Sine	SIN	=SIN(30)	0.5
Cosine	COS	=COS(15)	0.9659258
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.

Trigonometric function: The unit of angle θ is degree (°).

Arc tangent: -90 < 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.

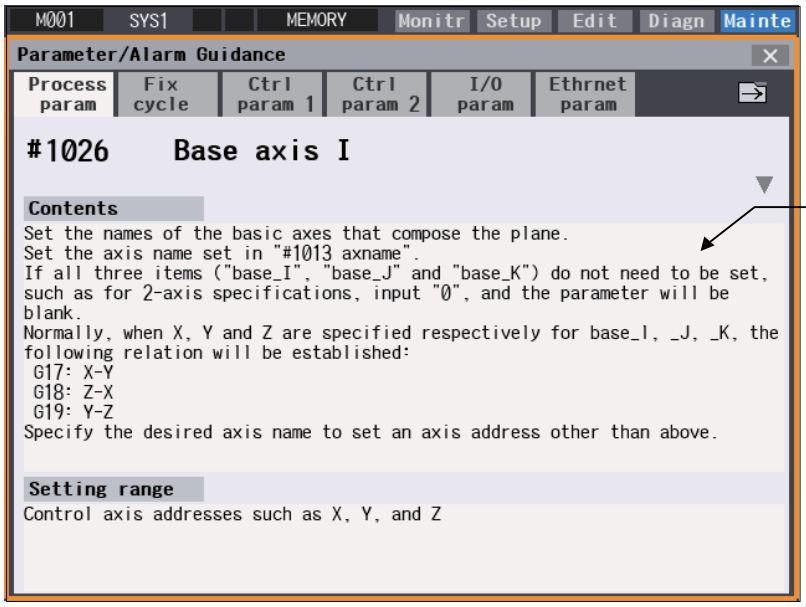
2.5 Guidance Function

2.5.1 Parameter Guidance

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

Screen Configuration

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



The parameter No., name, details and setting range are displayed.

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 parameter guidance (Example: "#8005 ZONE r")

(1) Press the menu [Param] on maintenance screen. ➡ The parameter screen is displayed.

(2) Press the [↓] key, and move the cursor to "#8005 ZONE r" parameter.

(3) Press the [?] key. ➡ The guidance window is opened, and the detail of "#8005 ZONE r" parameter is displayed.

Note

(1) The contents of the parameter "#8004 SPEED" (the previous parameter on the parameter screen) are displayed by pressing the PAGE key. However, when the PAGE key is pressed on the first parameter of each parameter type, the displayed contents do not change. The contents of the parameter "#8006 ZONE d" (next parameter on the parameter screen) are displayed by pressing the PAGE key. However, when the PAGE key is pressed on the last parameter of each parameter type, the displayed contents do not change.

Pressing the [↑] or [↓] key scrolls the explanation contents of the parameter guidance in a page unit.
When the [↓] key is pressed while the last line is displayed, next guidance appears. When the [↑] key is pressed while the top line is displayed, the previous guidance appears.

However, when the [↑] key is pressed while the first parameter guidance is displayed, or when the [↓] key is pressed while the last parameter guidance is displayed, the display does not change.

(2) The contents of the first parameter for the fixed cycle parameter are displayed by pressing the key. While the parameter guidance window is displayed, the displayed parameter contents are recorded for each parameter type. When the parameter type is changed with / key, the contents of the previously displayed parameter are displayed.

(3) When the parameter No. could not be acquired (when the cursor was non-display or was on a blank or comment line), the top of parameter guidance for each parameter appears.

(4) The parameters whose contents/setting range are similar or same are collectively displayed.
(Example: PLC integrated timer)

(5) For machining condition parameters on the machining condition setting screen, the cursor position does not move after opening and closing the parameter guidance.

Precautions

(1) If the [?] key is pressed when alarm occurs, the alarm guidance window will open.

(2) Once the guidance data is read, the guidance can be displayed even when the memory card is removed.

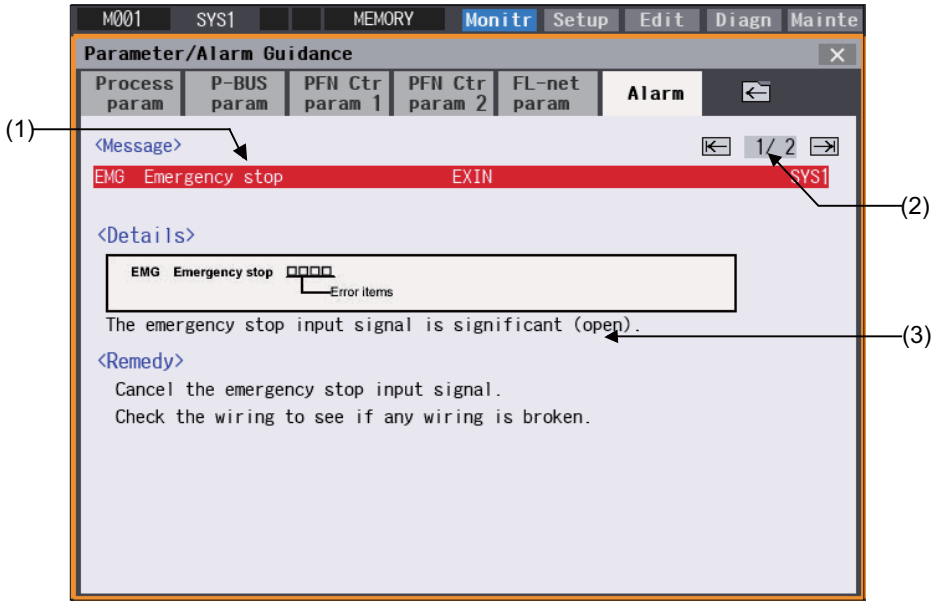
(3) Press the [?] key on a screen other than [Mainte] - [Param] and [Setup] - [User] when an alarm is not being occurred, and the head parameter guidance of parameters is displayed.

2.5.2 Alarm Guidance

The alarm guidance is the function that displays message, details and remedy for the currently occurring alarms and the alarms being displayed on the alarm history screen.

Screen Configuration

The alarm guidance is displayed in "Alarm" tab on "Parameter/Alarm guidance" window.
The "Alarm" tab 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 items	Details
(1) Messages	This displays "NC alarm" and "PLC alarm message" for the currently occurring alarm and the alarm being displayed on the alarm history screen. The displayed content is same as the content displayed in the alarm message of the diagnosis screen.
(2) Number of page	•When the guidance of currently occurring alarm is being displayed This displays "order of priority / total number of occurring alarms". •When the guidance of the alarm history is being displayed "Alarm number counted from the top/maximum number of the alarms which can be displayed" is displayed.
(3) Details/Remedy, etc.	This displays detail and remedy, etc. for alarm message.

Operation method

All alarms are displayed for alarm guidance regardless of part system.

If the [?] key or the  key is pressed on any screen when an alarm occurs or when the alarm history screen is displayed, alarm 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.

While the alarm history screen is being displayed, the guidance for the alarm displayed on the screen is displayed even if any other alarm occurs.

Displaying the alarm guidance

(1) Press the [?] key or the [?] key during occurring alarm on monitor screen.
(Example) When "H/W stroke end axis exists" and another alarm occur.

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

Note

(1) Pressing the [PAGE ▲] or [PAGE ▼] key scrolls the explanation contents by page.

(2) Pressing the [↑] or [↓] key scrolls the explanation contents in a line unit.

(3) When each parameter tab, such as the parameter, fixed cycle parameter, etc., is opened with the [←] key, the contents of first parameter in each parameter tab is displayed.

(4) The alarm guidance is executed for the alarm that occurs when the [?] key or the [?] key has been pressed. Therefore, the guidance is not displayed for the alarm that has been handled or has occurred while the alarm guidance is displayed.

(5) If the [?] key is pressed when an alarm is not occurring, the parameter guidance appears. In this state, if the "alarm" tab is selected with the [→] key, nothing is displayed in the alarm guidance panel. (These columns are blank.)

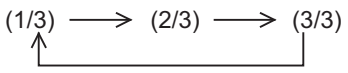
(6) All the explanations of the alarm having the same error class and No. are displayed in the details column.

(7) When no alarm or only one alarm occurs, the arrow next to the page number is not displayed.

Changing the alarm guidance

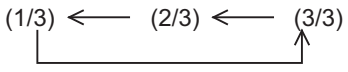
(1) Press the [→] key or [→|] key when the alarm guidance is displayed.

→ 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.
(Example) When 3 alarms occur.



(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.
(Example) When 3 alarms occur.



Note

(1) Up to 18 alarm or warning messages can be displayed for alarm guidance from highest priority.
(The messages are displayed in order of NC alarm or warning message (10 messages maximum) -> PLC alarm message (four messages maximum) -> NC alarm message for background check (four messages maximum).)

(2) Even if the [→], [←], [|←] or [→|] key is pressed when only one alarm occurs, the screen display is not switched.

Precautions

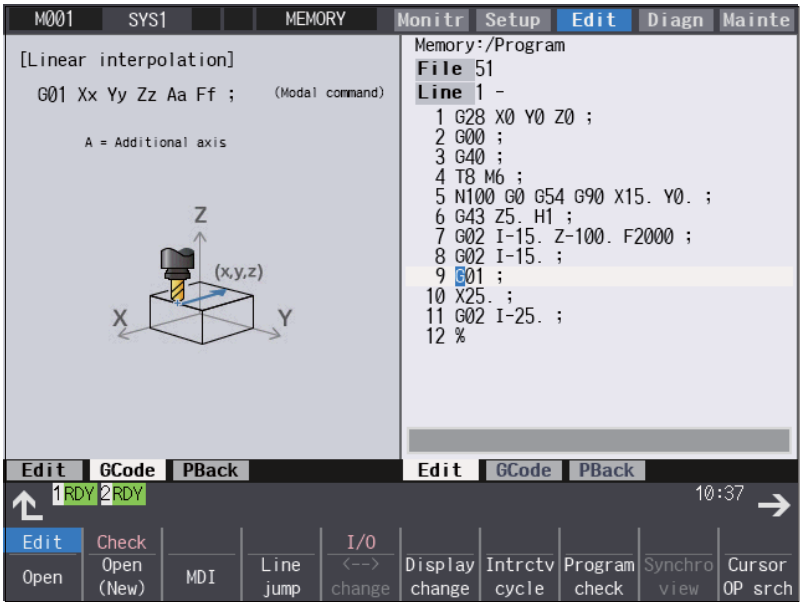
(1) The alarm guidance of stop code and operator message is not displayed.

(2) The PLC device information, which is displayed for the PLC alarm message guidance, displays the PLC device status when the alarm guidance is displayed. Accordingly, even if the device value changes when the alarm guidance is being displayed, the displayed PLC device information is not updated automatically. To update the information, close the alarm guidance once and display it again by pressing the [?] key again. However, when the PLC device information is redisplayed with [→], [←], [|←-] or [→|] key at occurrence of multiple alarms, the PLC device information is updated.

(3) Once the guidance data is read, the guidance can be displayed even when the memory card is removed.

2.5.3 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.



G code displayed in the G code guidance area is refreshed as the cursor moves across the program edit screen. When you change the available edit area during simultaneous program edit (3 programs), G code guidance is also refreshed.

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 is not refreshed because the cursor is out of the block.

G91 G17 ; → The guidance for G17 is displayed.

G91 G17 ; → The guidance for G91 is displayed.

G91 G00 X100. Y100. ; → The guidance for G00 is 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.

Gnnnnnnnn.m L ll ("n", "m", and "l" are arbitrary number.)

Commanded value	Number of valid digits	Remarks
n (integer)	1 to 8 digit	When including "0" before integer command value, unnecessary "0" is ignored.
m (decimal places)	0 to 1 digit	-
l (value commanded using "L")	0 to 2 digit	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 is not displayed even if the above condition is met.

G00 X100. (GCODE) ; → "G" included in the comment "GCODE" is ignored, thus, G code guidance is not refreshed.

N11 GOTO12 ; → "G" included in the command "GOTO" is ignored, thus, G code guidance is not refreshed.

G91 G120 X100. Y100. ; → G code "G120" is not included in the specifications, thus, G code guidance is not refreshed.

G00000031.1 ; → The guidance for G31.1 is displayed.

G00000031.10 ; → Commanded value exceeds the number of valid digits, thus, G code guidance is not refreshed.

L50 G10 ; → The guidance not for "G10 L50" but "G10" is displayed.

Precautions for G code guidance

(1) G code guidance may be refreshed when the cursor is located near the comment.

(Example) (PROG01)[;] → The guidance for G01 is displayed.

(2) When the cursor is located at the axis address, etc., the guidance for the nearest G code is displayed even if the address has no relationship with the G code.

(Example) G01 G90 X100[.] ; → G code that has a relationship with "X100." is "G01", however, G code guidance for "G90" is displayed.

(3) When moving the cursor to the block with no G code as follows, the contents of the G code guidance will differ depending on the cursor's route even if the cursor is in the same block.

(Example)

N01 G91 G01 X100. F1000[;]
↓
N02 [Y]100. Z100. [;] ← ... a
↑ b
N03 [G]02 X50. Y50. R50. ;

→ When the cursor is moved by the route "a", the guidance for G01 which is displayed last in N01 block is also displayed in N02 block.
When the cursor is moved by the route "b", the guidance for G02 which is displayed last in N03 block is also displayed in N02 block.

(4) The G code guidance description is common to M8 Series.
Thus, G code could not function though the guidance is displayed.

(5) Basically, the G code format in the G code 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 that has two or more meanings, both contents is displayed.

(8) G code guidance will not be refreshed while the character string is searched or replaced.

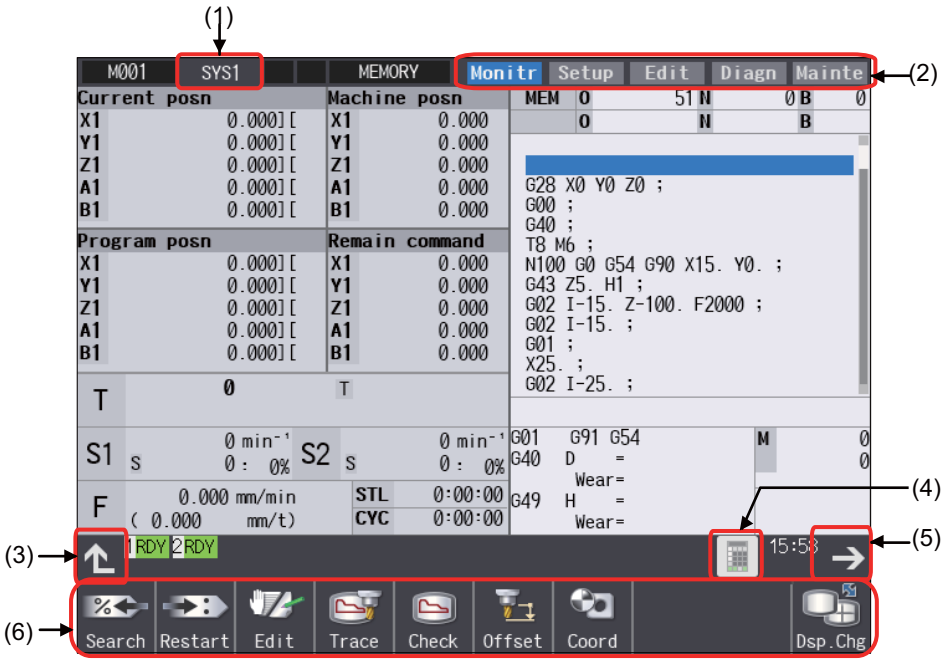
(9) G code guidance cannot be displayed in mass-editing.

(10) If "cmdtyp" varies in each part system for the L system, the display is switched to the guidance of the corresponding part system when the part system is changed during the G code guidance display.

2.6 Touchscreen Functions

Touchscreen operation is available with a touchscreen display.

2.6.1 Basic Operation



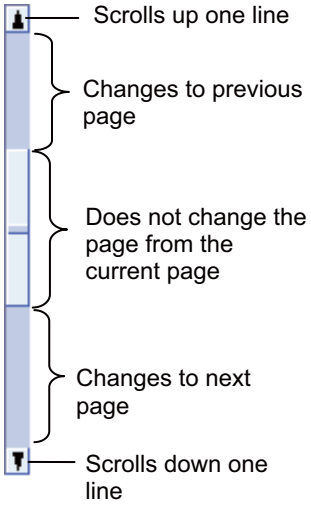
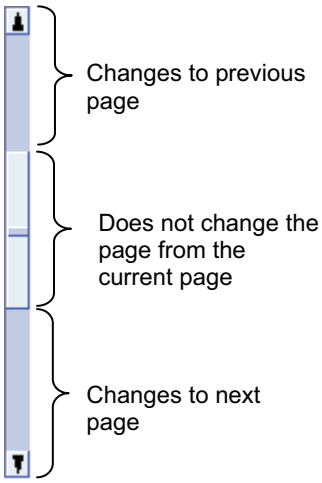
Display items

Display items	Details
(1) System change	This switches from the current part system to the next part system.
(2) Function change	A selected screen will open. As in the case with a screen change by key inputting, if a pop-up window is displayed, close it before changing screens. If there is a file under editing, confirm if the file is saved before changing screens.
(3) Menu operation cancel	The displayed screen operation menu is switched to the screen selection menu of the current screen group. This is also used to cancel the menu operations of the displayed screen.
(4) Software keyboard button	This displays the software keyboard on the screen.
(5) Menu change to the next	This switches to the next menu if available.
(6) Menu selection	When a menu is selected by touching, the menu's process is executed.

Page changing area by scroll bar

If you touch [▲] or [▼] of the scroll bar, the screen is scrolled up or down by one line. Also, if you touch the empty portion of the scroll bar, the page is switched to the next page or previous page.

Pages cannot be changed by dragging the slider.

<Normal condition>	<Workpiece coordinate system offset window, W coordinate frame local variables, or T code list> (*1)
	

(*1) For the workpiece coordinate system offset window and local variable window on the monitor screen as well as coordinate system offset frame and T code list window on the setup screen, the page changes to the previous page when [▲] of scroll bar is touched, and the page changes to the next page when [▼] of scroll bar is touched. (It is not possible to scroll in a line unit.)

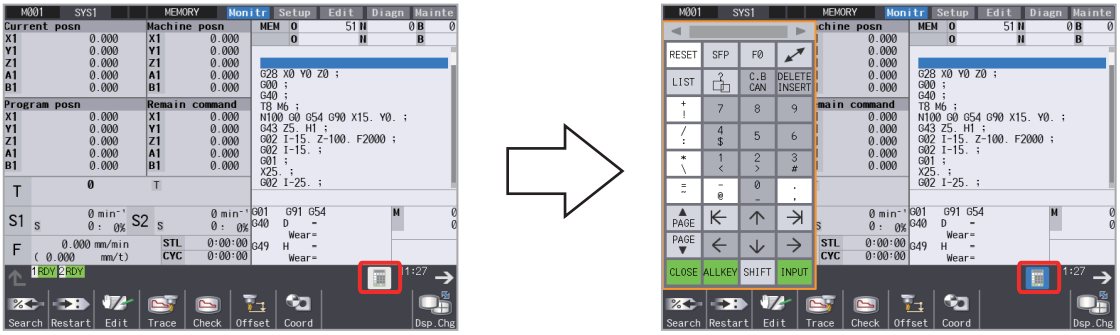
Disable touchscreen operation

When the touchscreen needs to be disabled, set the parameter "#1480 tp_invalid" (Disable touch panel operation) to "1", or turn the touchscreen operation invalidation signal ON. Refer to the instruction manual issued by the MTB for details.

2.6.2 Software Keyboard

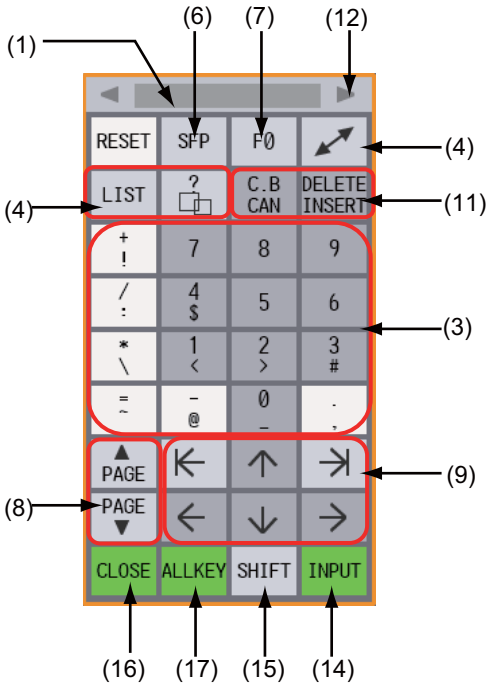
With the software keyboard, a key input is possible by a key menu on the software keyboard which is displayed on the screen. Software keyboard is displayed if button is touched in each screen. In this case, screen or menu status will not change. Software keyboard is always displayed on the front screen. When the [CLOSE] key on the software keyboard is touched, the software keyboard will be closed.

Software keyboard is enabled when the parameter "#11010 Software keyboard" is set to "1", "2" or "3". Even if the parameter is "0", software keyboard can be used when the keyboard has not been connected. When software keyboard is disabled, button is not displayed.

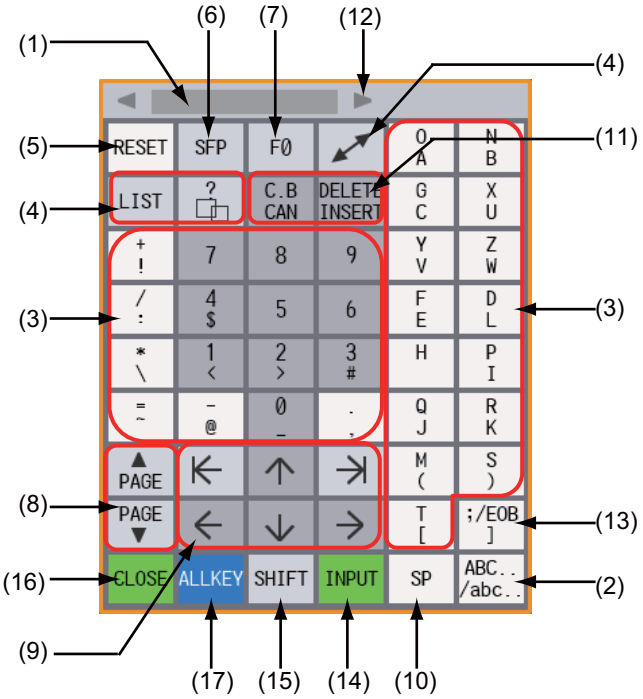


During the software keyboard is displayed, the button is inverted.

[Numerical keypad display]



[All keys display]



Display items

Display items	Details
(1) Input section	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 keys	This enters characters in the input area or screen.
(4) Window operation keys	[LIST]: This displays the menu list window. [?]: This displays the parameter/guidance window. [SHIFT] key + [LIST]: Switches the prototype screen and mass production screen. [SHIFT] key + [?]: Displays the custom screen (*1).  : This changes active windows.
(5) RESET key	This resets the NC.
(6) SFP key	This displays the custom screen (*1).
(7) F0 key	This displays the onboard screen or custom screen (*1).
(8) Page up/down keys	This displays the contents in the previous/next page. When the numerical keypad display keyboard is displayed automatically at the time of Yes/No confirmation, [Y]/[N] keys are displayed instead of page up/down keys.
(9) Cursor keys	If data is set in the display items on the screen, the cursor is moved to vertically/horizontally.
(10) Blank key	This enters a blank.
(11) Data correction keys	[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. [DELETE]: This deletes a letter before the cursor position. [C.B]: This cancels the settings in the input area. [CAN]: This cancels.
(12) Display position switch	This switches the display position of the software 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 the data setting key. Once [SHIFT] key is selected, it will be valid until another key is pressed or the software keyboard is closed. Menus are displayed in highlight while [SHIFT] key is valid.
(16) CLOSE key	This closes the software keyboard.
(17) Key display switch key	This switches numerical keypad display/all keys display.

(*1) Operations vary depending on the MTB specifications.

Input area of software keyboard

If the software keyboard is displayed while the input area exists on the screen, the input area of the software keyboard is displayed in the enable state. When the data setting key or data correction key is selected while the input area is enabled, enter a key in the input area of the software keyboard, and use the [INPUT] key to enter data in the screen.
Furthermore, if the software keyboard is displayed to edit a program file when the input area does not exist on the screen or while the editing window or MDI editing window is being displayed, the input area of the software keyboard is displayed into the disable state. In this case, directly enter a key in the screen.
When you enter a key to type the password, an asterisk "*" appears in the input area of the software keyboard.

Displaying/closing the software keyboard

Touching the  button on the lower left of the screen allows you to display the software keyboard on the screen.

(1) Touch the  button. → The software keyboard is displayed on the front screen. The button is highlighted.

(2) Touch the [CLOSE] key.
Or, touch the  button again. → The software keyboard will close.

Switching the view mode (all keys display/numerical keypad display)

Touching the [ALLKEY] key on the software keyboard allows you to switch the software keyboard view mode between "numeric keypad view" and "all key view". "Numeric keypad view" is specified by default.

(1) Touch the  button. → The software keyboard is displayed on the left side of the screen.
The software keyboard is displayed as a numerical keypad.

(2) Touch the [ALLKEY] key at the bottom of the software keyboard. → All keys, including alphabet character keys, are displayed.
The [ALLKEY] key is highlighted.

(3) Touch the [ALLKEY] key at the bottom of the software keyboard. → The display is switched to the numerical keypad only.
The [ALLKEY] key is unhighlighted.

Changing the display position

The software keyboard can be relocated using one of the following methods:

- Touch the ◁ or ▷ key at the top of the software keyboard.
  The software keyboard moves to the three fixed positions: left, middle or right.
- Drag or flick on the software keyboard
  Drag the screen (touch the surface and move your finger vertically/horizontally without losing contact) or flick the screen (quickly run your fingertip vertically/horizontally across the screen) to move the keyboard to any desired position.

■Touch the ◁ or ▷ key at the top of the software keyboard.

(1) Touch the button. → The software keyboard is displayed on the left side of the screen.
The button is highlighted.

(2) Touch the ▷ key located at the upper right corner of the software keyboard. → The software keyboard moves to the right, and it appears on the center of the screen.

(3) Touch the ▷ key located at the upper right corner of the software keyboard. → The software keyboard moves to the right, and appears on the right side of the screen.

Note

(1) When the ◁ key is pressed, the software keyboard moves to the left.

■Drag or flick on the software keyboard

(1) Touch the button. → The software keyboard is displayed on the left side of the screen.
The button is highlighted.

(2) Drag or flick in the lower right direction on the software keyboard. → The software keyboard moves in the lower right direction.

(3) Touch the ▷ key located at the upper right corner of the software keyboard. → The software keyboard moves to the right, and appears on the right side of the screen.

Entering data when the input area is enabled

If displaying the software 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 software keyboard input area will be valid. At this time, the key selected on the software keyboard is temporarily displayed in the software 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 tab on the setup screen and touch the  button. → The software keyboard is displayed on the left side of the screen.
Software 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 software keyboard input area.

(4) Touch the [INPUT] key. → "15" is input at the tool compensation amount cursor position.

Note

(1) It is possible to make an operation input in the entry field of the software keyboard. For details on settings and operations, refer to "2.4.2 Inputting Operations".

(2) If 20 or more characters are entered in the entry field of the software keyboard, character strings displayed in the input area are scrolled in sequence.

(3) If you touch the [←] or [→] key while the input area of the software keyboard is enabled, the cursor will move in the relevant direction in the entry field.

Entering data when the input area is disabled

If software 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, software keyboard input area will be invalid. At this time, the key selected on the software keyboard is directly input to the screen.

(1) Display the edit window on the monitor screen and touch the  button. → Software keyboard is displayed.
Software 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.

2.6.3 Touchscreen Operation Functions

The terms used in the function descriptions here are defined as follows.

Term	Explanation
Flick	A quick sweeping motion made while touching the screen
Drag	A gesture made by placing your finger on the screen and moving it across the screen without losing contact
Pinch-in	A gesture made by pinching inward between the thumb and index finger around an object on the screen
Pinch-out	A gesture made by spreading the thumb and index finger on the screen
Tap	A light tap of a finger on a particular location on the screen
Double tap	Two light taps of a finger on a particular location on the screen
Hold	To tap and hold a finger on a particular location on the screen
Grab	To grab with multiple fingers touching the screen

The following operations are available using the touchscreen. Refer to the operation description of each item below for details.

Operation	Reference
Changing the Menu Display	2.6.3.1
Displaying the Menu Selection List	2.6.3.2
Scrolling the Displayed Data	2.6.3.3
Changing a Page	2.6.3.4
Switching to the Next Axis or Part System	2.6.3.5
Selecting a File	2.6.3.6
Displaying Edit Window/Selective Display Window	2.6.3.7
Moving a Window	2.6.3.8
Changing the Font Size	2.6.3.9
Displaying the Parameter Guidance	2.6.3.10
Displaying the Alarm Guidance	2.6.3.11
Enlarging or Reducing, Moving, and Rotating a 3D Check Shape	2.6.3.12
Displaying the Screen List	2.6.3.13

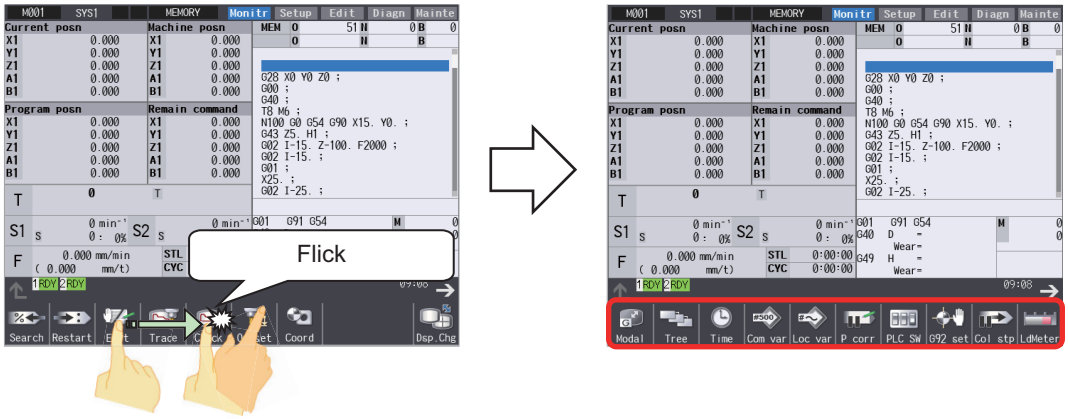
Touch operation enabled judgment

Touch operation starts when the valid range is touched, and remains active until your fingertip leaves the screen.
When touching multiple points, touch with all fingers withing the valid range and then move your fingertips.
The number of valid points for a touch gesture will vary by operation. Refer to the explanations of each screen for more details.

2.6.3.1 Changing the Menu Display

If the menu is flicked horizontally, the menu will slide and change to the previous or next menu.

(Example)



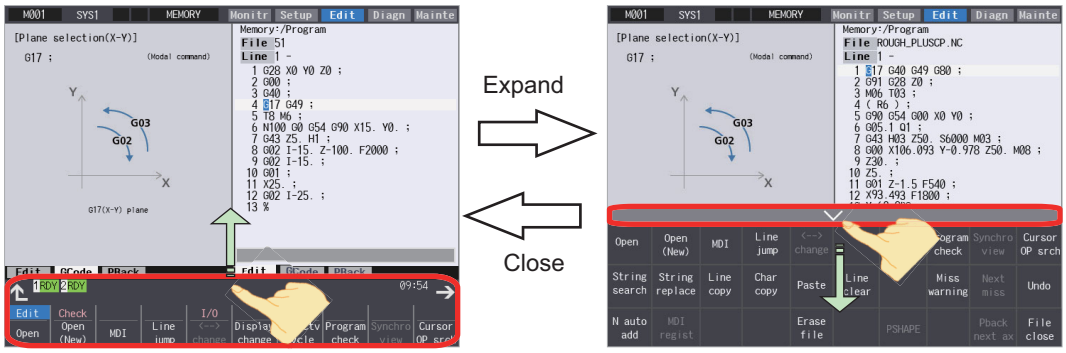
2.6.3.2 Displaying the Menu Selection List

Drag or flick the menu button up to display the menu selection list.

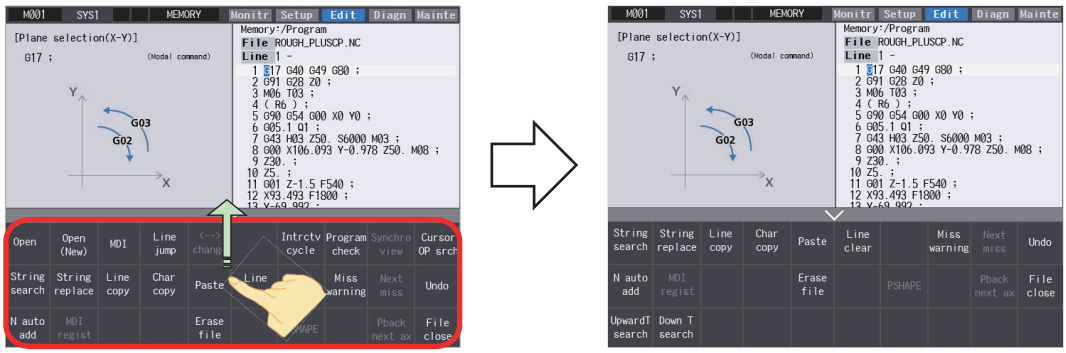
When the menu selection list is displayed, it can be hidden by dragging the down arrow ( ) down.

You can also drag or flick the menu selection list up or down to scroll through the list and display the remaining menu buttons that are not visible.

(Example) Dragging to display/hide the menu selection list

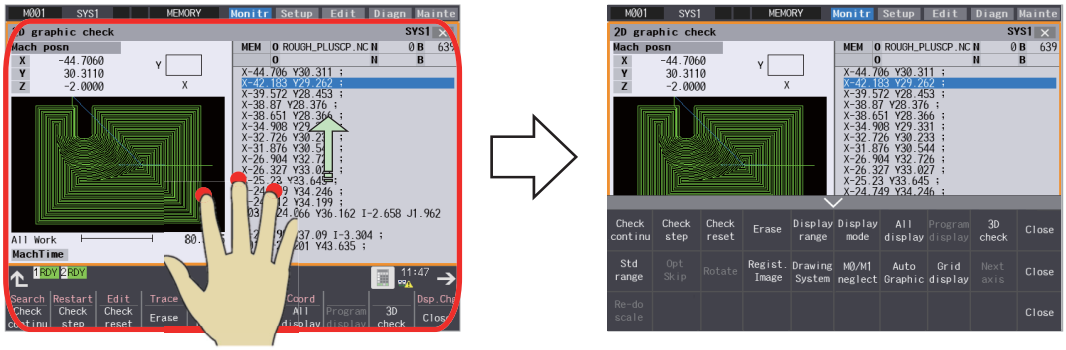


(Example) Dragging to scroll through the menu list



The menu selection list can also be displayed by dragging upward with three or more fingers outside the display area of pop-up windows and the software keyboard. With full-screen pop-up windows, the gesture can be made within the pop-up window display area.

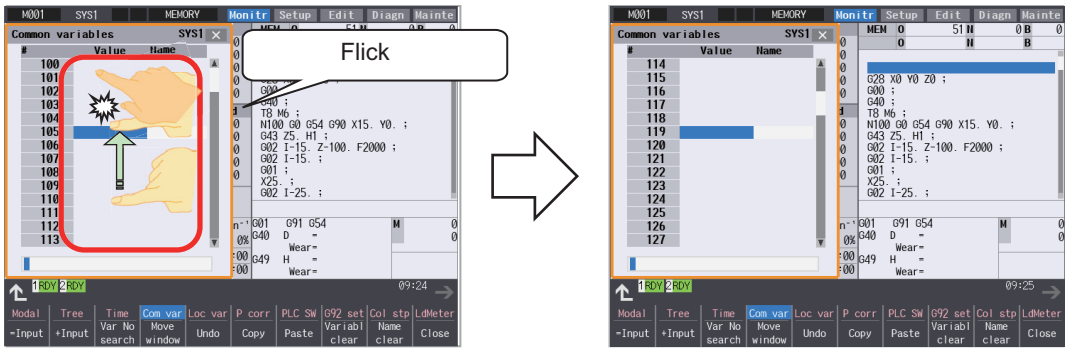
(Example) Displaying the menu selection list by dragging upward with three fingers



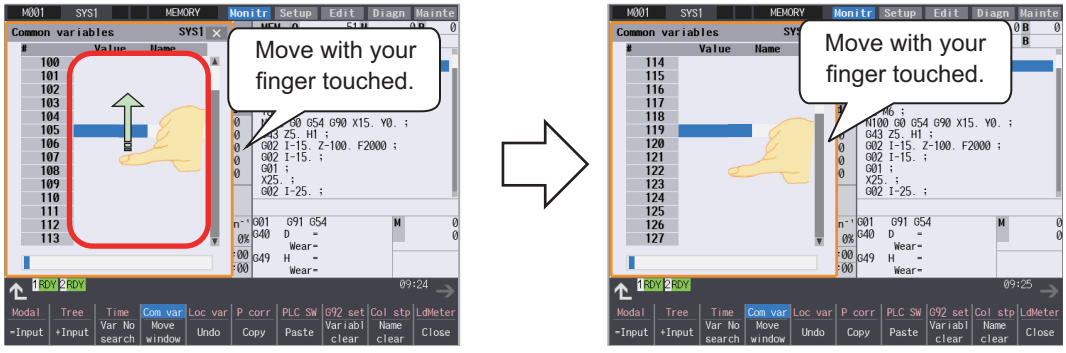
2.6.3.3 Scrolling the Displayed Data

By flicking or dragging the screen vertically to edit the monitoring, setup, or editing program or process various table data items such as the compensation amount and common variables, the displayed data is scrolled.

(Example) Scroll by vertical flick



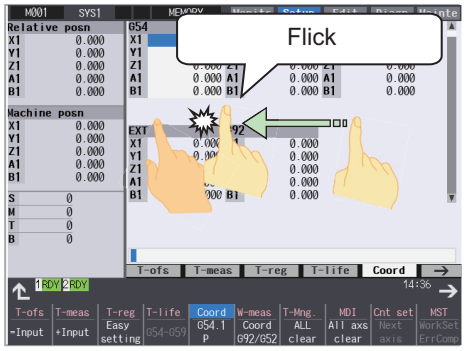
(Example) Scroll by vertical drag



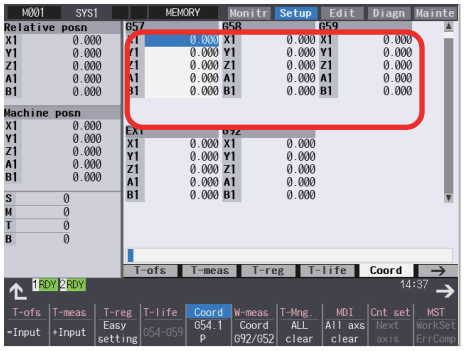
2.6.3.4 Changing a Page

On screens where the page key is used to switch the appearance of the entire screen, such as the workpiece coordinate system offset and parameter guidance screens, flick horizontally to switch to the previous or next page.

(Example) Page changing for workpiece coordinate system offset

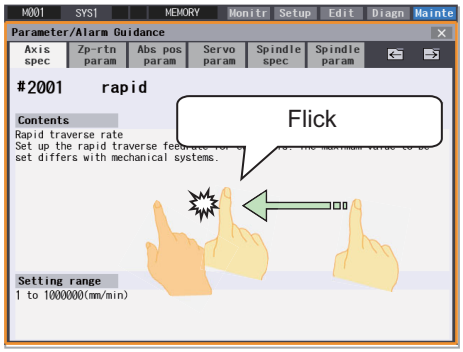


A workpiece coordinate system offset (G54-G56) is being displayed.

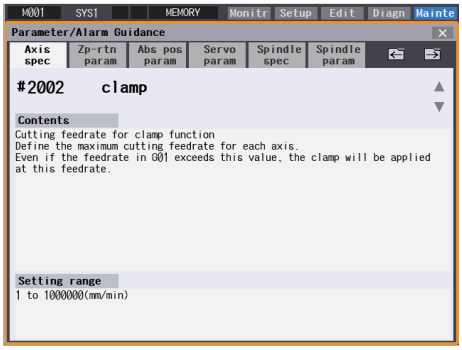


Changed to the next workpiece coordinate system offset (G57-G59) page.

(Example) Page changing for parameter guidance



A parameter guidance (#2001) is being displayed.

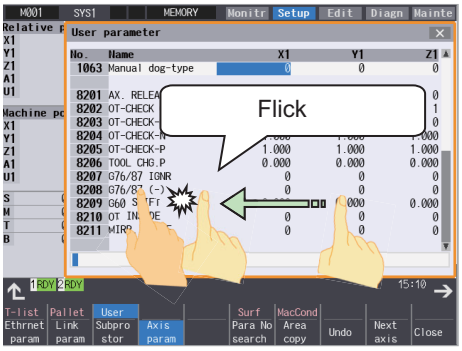


Changed to the next guidance (#2002) page.

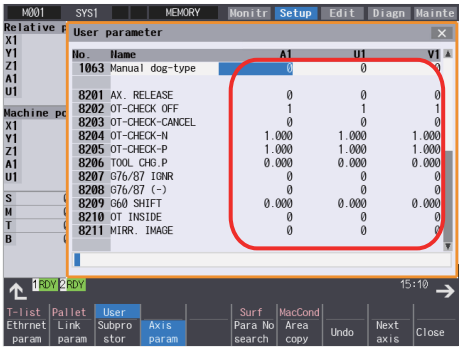
2.6.3.5 Switching to the Next Axis or Part System

On screens such as the axis parameter screen where pressing [Next axis] will switch all displayed axes, flick horizontally to switch an axis or a part system to be displayed.

(Example) Displaying the next axis of the axes' parameters.



The axis parameters (X, Y and Z axes) are displayed.

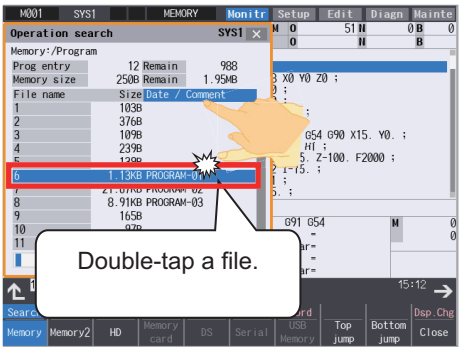


The next axes (A, U and V axes) are displayed.

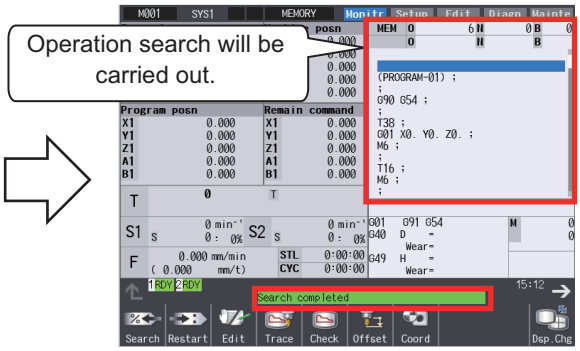
2.6.3.6 Selecting a File

If you double-tap a target file on a screen that displays a file list, for example, on the Operate search screen, operation is performed in the same way as when the [INPUT] key is pressed.
In addition to the following examples, files can also be selected on the [Edit] - [Open] screen and on the [Mainte] - [InOut] screen.

(Example) Double-tap on [Monitr] - [Search]



The Search screen is being displayed.



Search is executed.

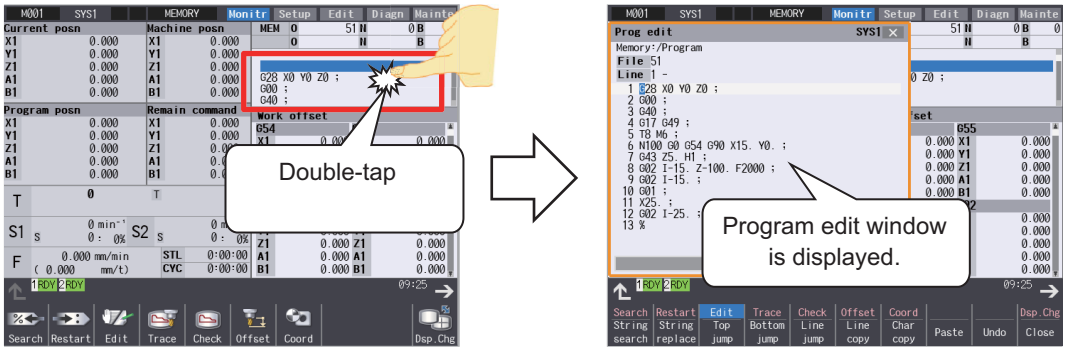
Note

(1) The subsequent operation varies depending on the target you double-tapped.
When the target is a file: The selected file name is specified.
When the target is a folder: The list of files in the selected folder is displayed.

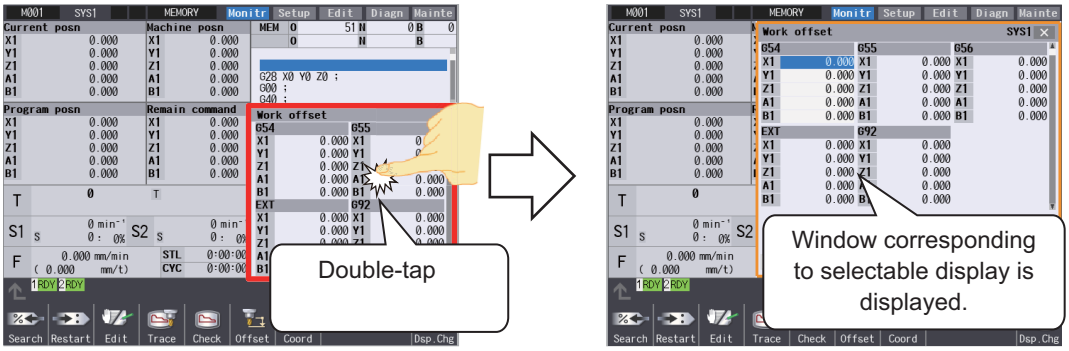
2.6.3.7 Displaying Edit Window/Selective Display Window

If you double-tap the buffer display area on the operation screen, the program being searched is displayed on the edit window. You can double-tap the buffer display area for each part system during 2-part system/3-part system simultaneous display.
Also, if you double-tap the selective display area, the window which you can operate the menu assigning to the selective display, in the same manner when you press [INPUT] key.

(Example) Displaying the edit window by double-tapping the buffer display area



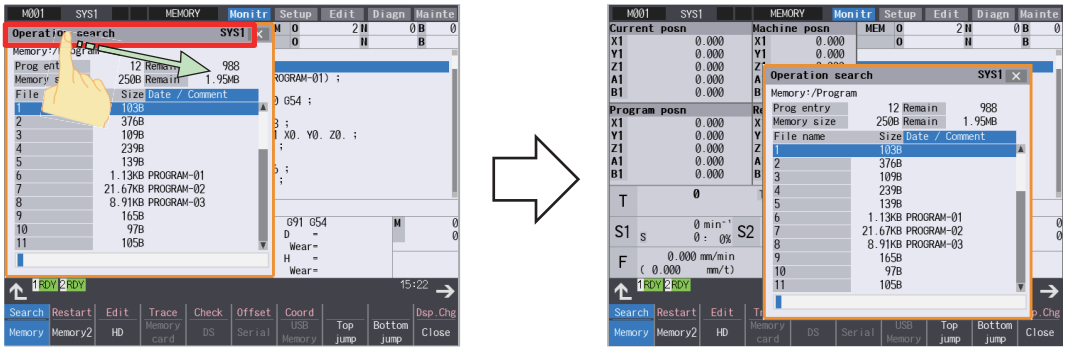
(Example) Displaying the window by double-tapping the selective display area



2.6.3.8 Moving a Window

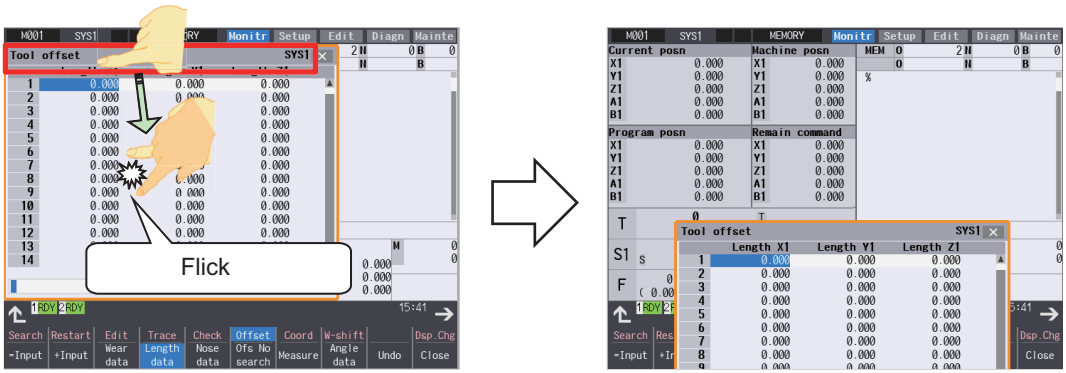
On the search screen and other window display screens, the window can be moved to the desired position by flicking or dragging the title bar with a single finger. Similarly, a window can also be moved by touching three or more fingers within the window and flicking or dragging. The window is not moved during file searching, etc.
The window display position after move is maintained until the power is turned ON again.

(Example) Window movement on [Monitr] - [Search] screen (Drag)



Before window move　　After window move

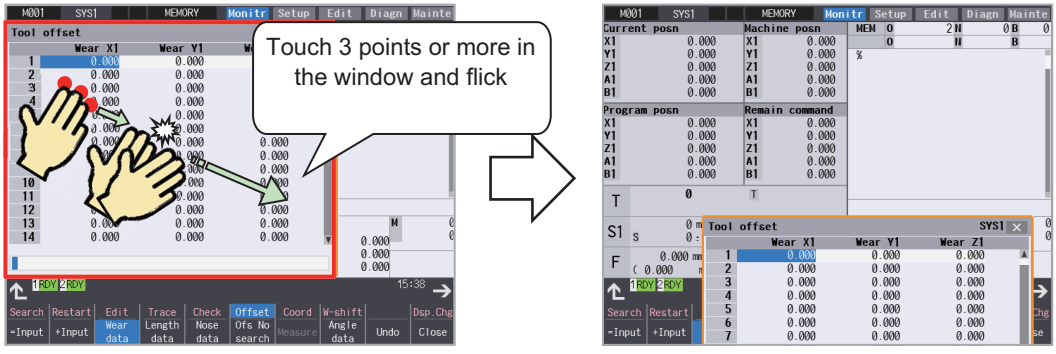
(Example) Window movement on [Monitr] - [Offset] screen (Flick)



Before window move　　After window move

If more than two points within the window area are touched, the window can be moved by flicking or dragging it.

(Example) Window movement on [Monitr] - [Offset] screen (Flick)

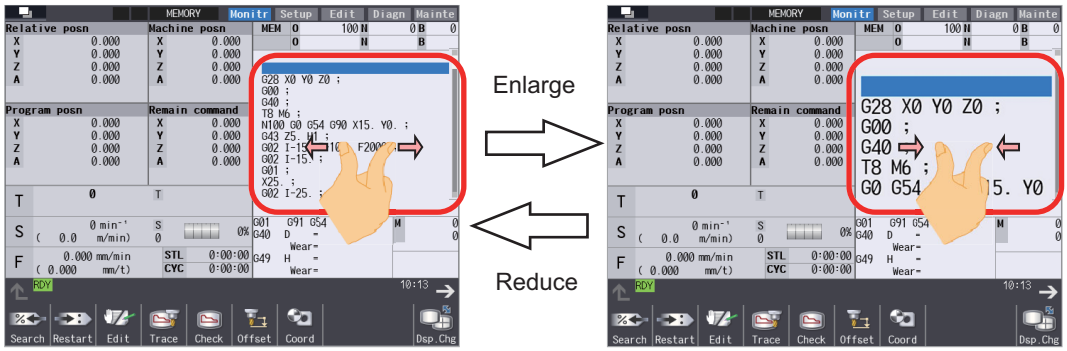


Before window move　　After window move

2.6.3.9 Changing the Font Size

In the buffer, program, and file list display areas, pinch in to reduce the font size or pinch out to enlarge the font size.

(Example) Adjusting the font size of the buffer display area



In addition to the above screens, the font size can also be enlarged or reduced on the following screens:

- File list area in the monitor/edit screen
- Edit window in the monitor/setup/edit screen
- Program display area and input/output windows in the edit screen

Note

(1) This function cannot be used in screen layouts with buffer display areas of three or fewer lines, such as the following screens:

4-part system simultaneous display screens, quad counter display screens, 2-part system simultaneous display screen with counter 1 (expand)

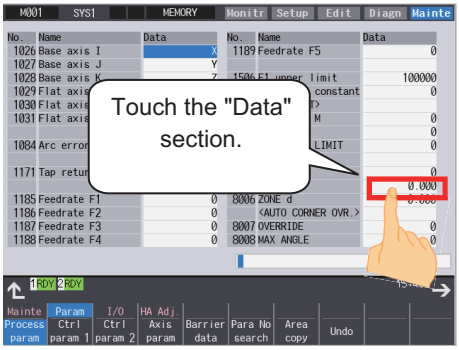
(2) Font sizes for the monitor screen edit window, setup screen MDI edit window, and edit screen are linked and change as a group. However, if the maximum font sizes are different, the font size will not change.

(3) Font sizes for the file list on the I/O window of the maintenance screen cannot be reduced or increased by pinching in or out.

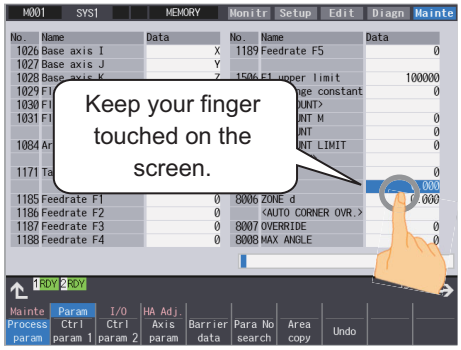
2.6.3.10 Displaying the Parameter Guidance

Holding down the parameter data area on the parameter screen of the [Setup] or [Mainte] (Maintenance) screens will display guidance for the parameter.
The length of time which is recognized as a long press can be set with the parameter "#8995 Touchop longtaptim".

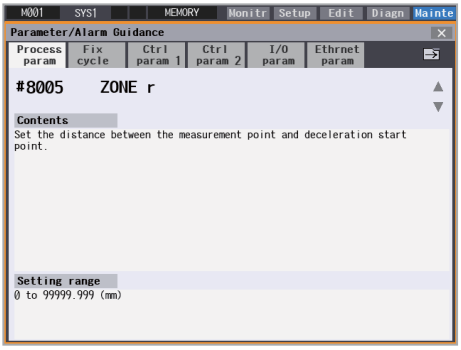
(Example) Displaying #8005 guidance



The parameter screen is being displayed (the cursor position is at #1026).



The cursor position moves to #8005.



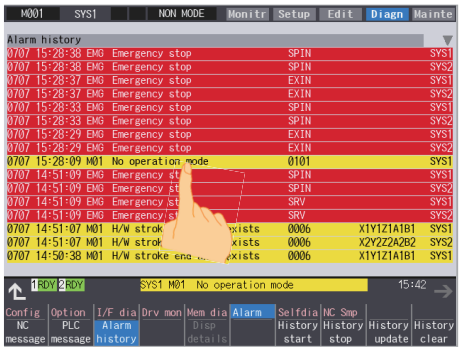
#8005 guidance is displayed.

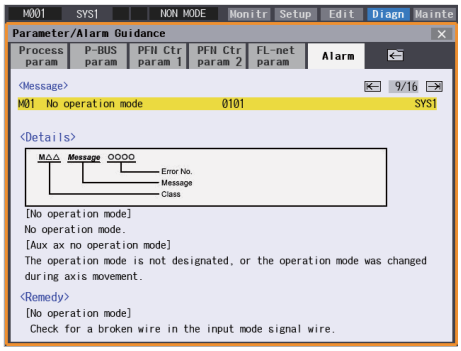
Note

(1) Do not remove your finger until the guidance is displayed.

2.6.3.11 Displaying the Alarm Guidance

If you hold down the alarm data area on the alarm history screen of the [Diagn] (Diagnosis) screen, the guidance of the alarm is displayed.
The length of time which is recognized as a long press can be set with the parameter "#8995 Touchop longtaptim".

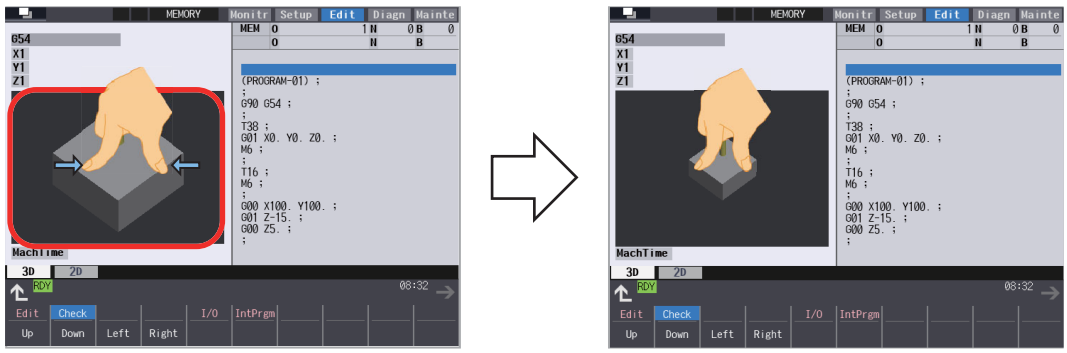




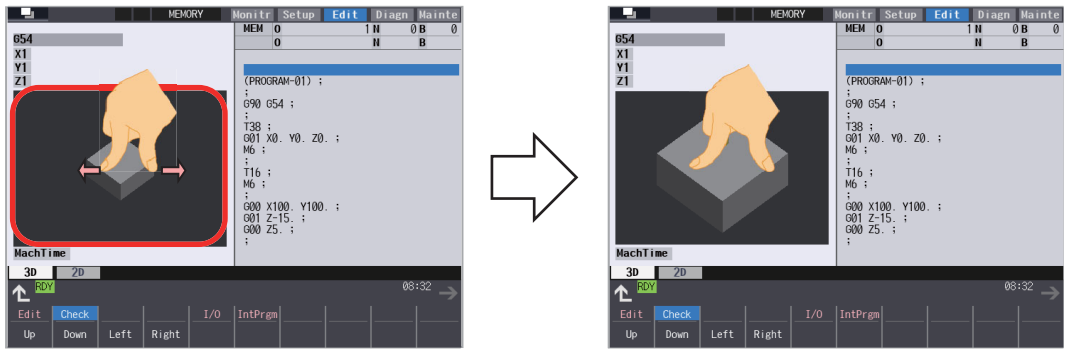
2.6.3.12 Enlarging or Reducing, Moving, and Rotating a 3D Check Shape

Pinch in to reduce and pinch out to enlarge the machining shape for 3D check. Also, if you drag the screen, you can move or rotate a machining shape. However, touch operations on the 3D check shape are disabled during check searching or while the tool setting screen or "Work set" window is being displayed. Touch operations on the 3D check shape are enabled while the software keyboard is being displayed.

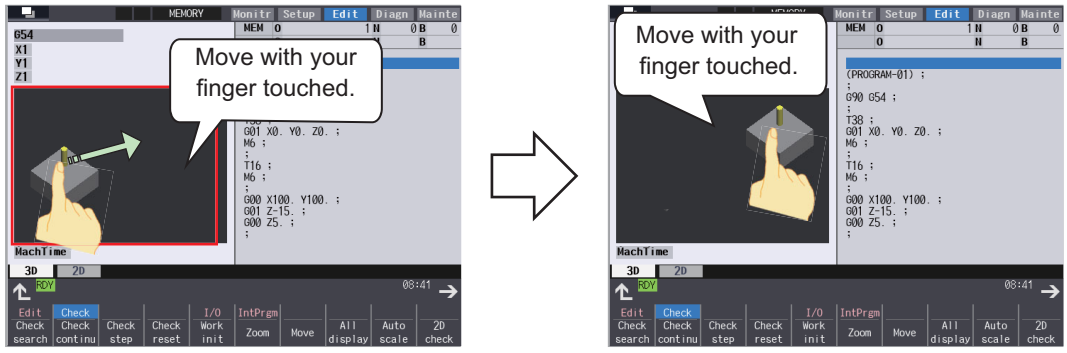
(Example) Zoom-out



(Example) Zoom-in

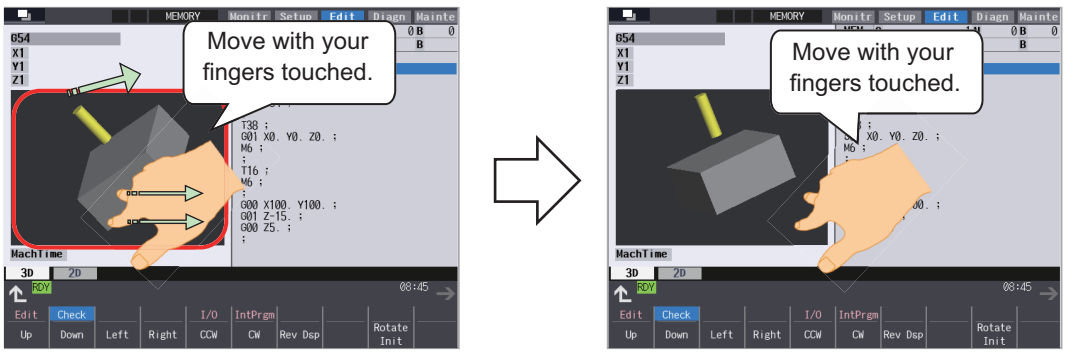


(Example) Move

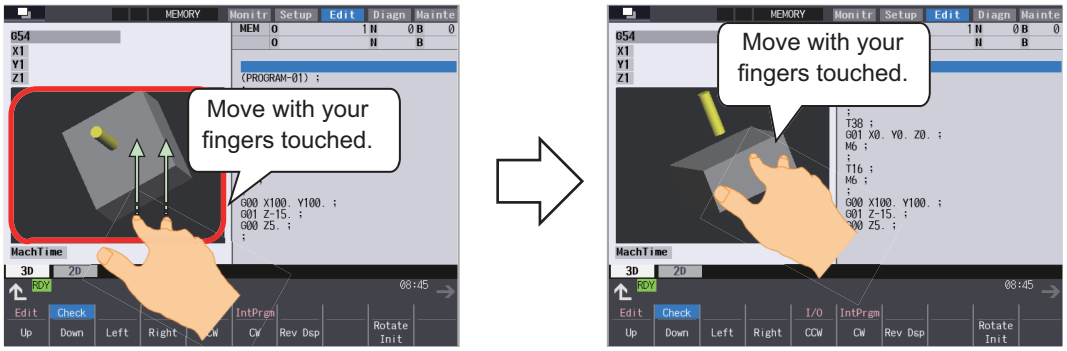


The machining shape can be rotated by dragging it with two fingers.

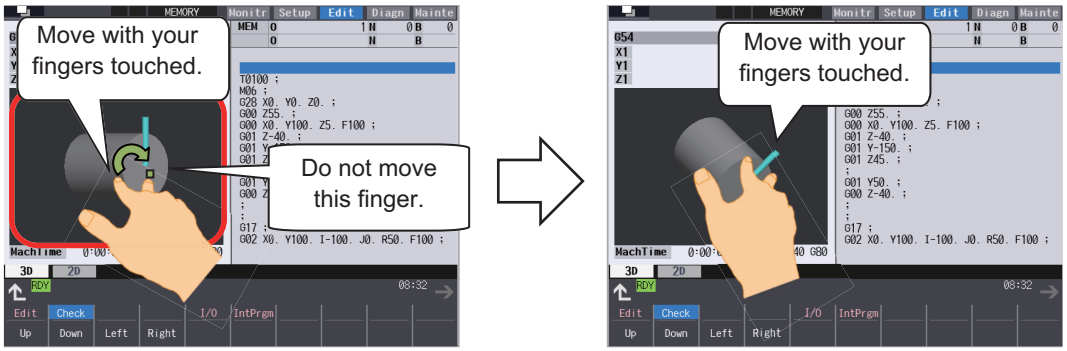
(Example) Rotate about the horizontal axis



(Example) Rotate about the vertical axis



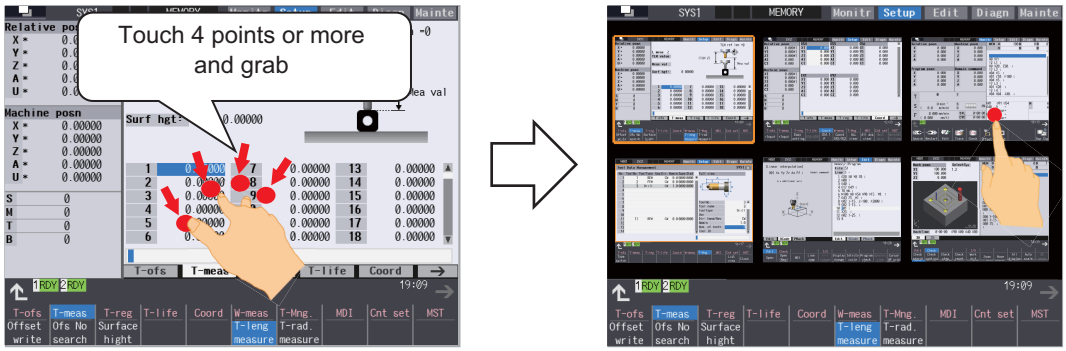
(Example) Rotate about the center coordinate of the material



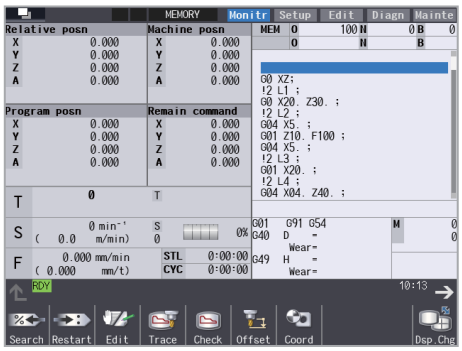
2.6.3.13 Displaying the Screen List

Grabbing the screen at four or more points will show a list of the previous screens displayed.
Touch any of the screens in the screen list, or select the screen with the ([←], [→], [↑], and [↓]) arrow keys and press the [INPUT] key to switch to the selected screen.
Some screens will not appear in the screen list when the screen is grabbed. Custom screens added by MTB may also not appear in the screen list.

(Example) Grabbing to switch screens



A list of the last 6 displayed screens is shown.



The selected screen is displayed.

Note

(1) The screen list will not display while the menu list, guidance, or alarm display windows are displaying, or when grabbing while a program is being edited.

(2) If a grab is performed while editing in the program edit window and the screen list is displayed without saving the program due to N or another key being pressed, selecting the program edit window again will return to the program edit window with the edit still in progress.

(3) If the screen selected from the screen list view is not registered in the main menu, the screen will change to the standard screen of each monitor, setup, edit, diagnose, or maintenance screen.

(4) The following screens will appear as individual screens on the screen list, but only the last screen displayed can be accessed from the screen list.

- ♦The 2D/3D check windows of the monitor screen
- ♦The 2D/3D check windows of the edit screen
- ♦The NAVI MILL/NAVI LATHE windows of the edit screen

For example, selecting the 2D check window from the screen list may switch to the 3D check window.

(5) If the parameter "#1759 cfgPR09/bit0" is set to "1", the screen list is not displayed even when the screen is grabbed.

2.7 Menu Customization Function

You can freely relocate the main menus of the monitor (Monitr), setup (Setup) and edit (Edit) screens.
This function enables to place the often-used keys in the first page.
It can be used when a parameter "#11032 Menu sel para lkof" (Validate menu selection parameter setting) is set to "2".
MTB password is required when it is set to "1".
The main menu of monitor screen, setup screen and edit screen correspond to the following parameters:

<Menu selection parameter>

Main menu for monitor screen: "#10501 Monitr main menu 1" to "#10540 Monitr main menu 40"

Main menu for setup screen: "#10551 Setup main menu 1" to "#10580 Setup main menu 30"

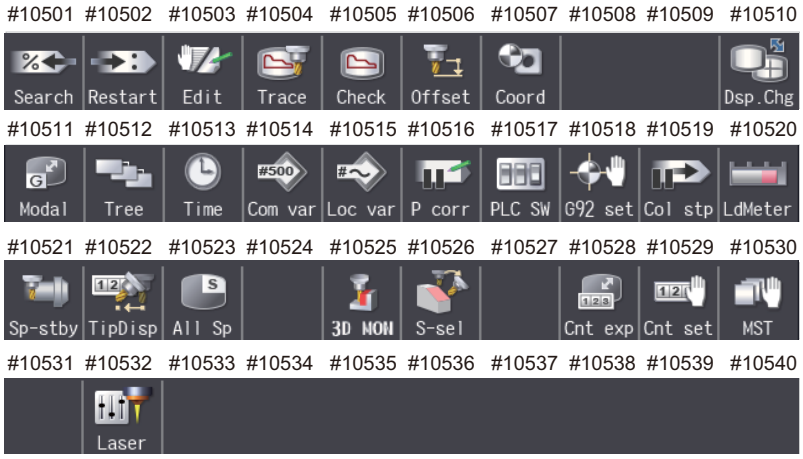
Main menu for edit screen: "#10601 Edit main menu 1" to "#10630 Edit main menu 30"

Refer to "Alarm/Parameter Manual" for details.

Setting example

Main menu for monitor screen is changed as follows.
The figure below is the default layout for M system.

<Default layout>



(1) Set the parameters "#10501 Monitr main menu 1" to "#10540 Monitr main menu 40" as shown in the right.

<Parameter> <After parameter change>

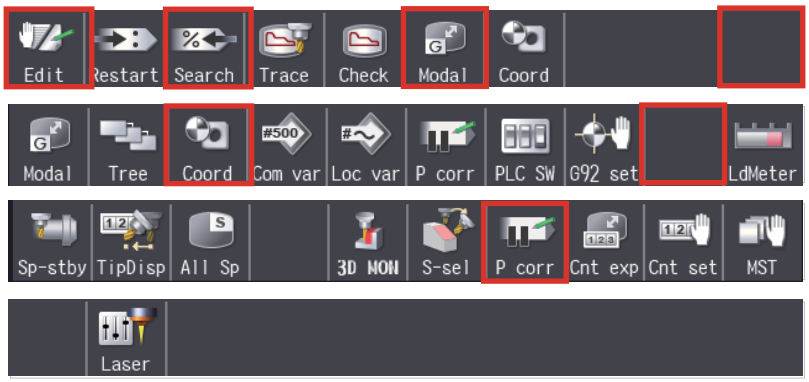
#10501	3 (Edit)
#10503	1 (Search)
#10506	11 (Modal)
#10510	-1 (Not display)
#10513	7 (Coord)
#10519	-1 (Not display)
#10527	16 (P corr)

Set "0" to the parameters other than the listed above.

(2) Turn the power ON again.

The new layout of the monitor screen's main menu is displayed.

<After parameter change>



Menus on page 1

Menus on page 2

Menus on page 3

Menus on page 4

Precautions

(1) When you set the number which the main menu is not allocated with the menu selection parameter, the menu is not displayed. Menu which is set with other menu non-display parameter ("#8923 Hide Edit-IO menu", "#8932 Hide measure scrn") is also not displayed even if it is set with the menu selection parameter.

(2) A menu selection parameter setting will be reflected to the menu list and menu selection list.

(3) Menus which are allocated to the short cut key (it is enabled after the axis address is input), such as [Counter Set] and [MST], become valid by pressing a short-cut key even if they are set as "Not display".

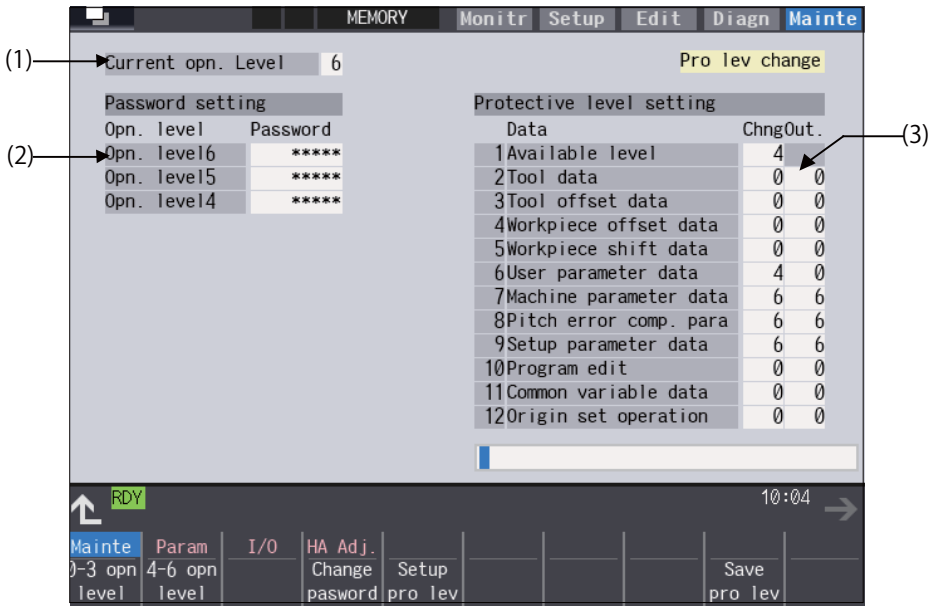
(4) When the parameter "#11032 Menu sel para lkof" (Menu select param setting enabled) is set to "0", the menu [Menu select] is not displayed.

(5) When the parameter "#11032 Menu sel para lkof (Menu select param setting enabled)" is set to "1" and the MTB password is not entered, the menu [Menu select] appears, but the menu select param setting screen does not appear even if you press the menu.

2.8 Changing the Operation Level (Protect Setting Screen)

Mainte – 

Operating the maintenance screen reduces a phenomenon in which defective workpieces run out due to an operation mistake, by implementing a restriction with seven operation levels. The operation level varies depending on the MTB specifications. Refer to the instruction manual issued by the MTB for details.



Display items

	Display items	Details
(1)	Current operation level	Displays the current operation level.
(2)	Passwords at operation levels 4 to 6	Displays the passwords of operation levels 4 to 6 with "*****".
(3)	Protection level of each data item	Displays the protection level of each data item. The data set below the current operation level can be changed as needed. Changing the protection level displays the message "Pro lev change" at the upper right. Change: Specify the protection level to change data (including a case to input a file). Output: Specify the protection level to output a file.

2.9 Assigning Arbitrary Character String to a User-defined Key

To use this function, define the registration number of the user-defined key and any character string, which is input when the user-defined key is pressed, in the configuration file (KEYDEF.TXT), then place the configuration file in the "M01:\MSYS\CUSTOM" folder.
There are two definition types of user-defined keys: type 1 and type 2. Specify the desired type with the parameter "#8956 User key type".
For information on the correspondence between the assignable user-defined keys and registration numbers, refer to "Key correspondence table" shown below.

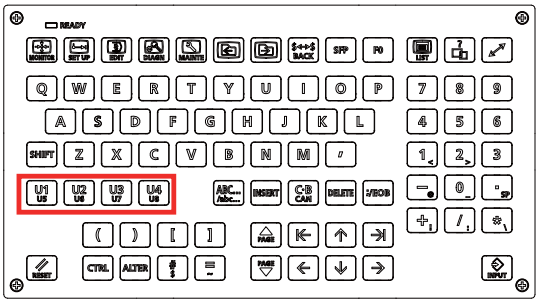
[Definition format]

n[character string] (n: Indicates a registration number between "0" and "7".)
Key correspondence table

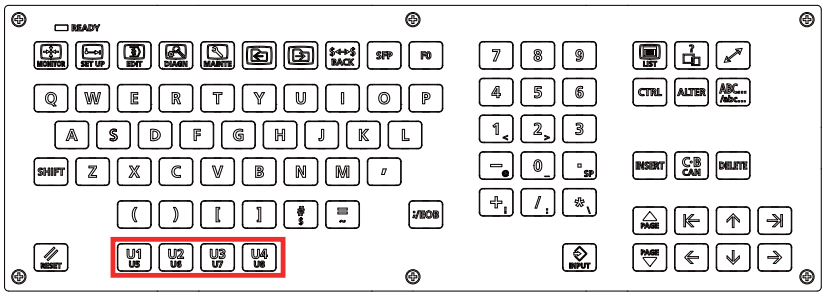
n	Key input
0	U1
1	U2
2	U3
3	U4
4	U5(SHIFT + U1)
5	U6(SHIFT + U2)
6	U7(SHIFT + U3)
7	U8(SHIFT + U4)

[Position of user-defined keys]

FCU8-KB047



FCU8-KB083



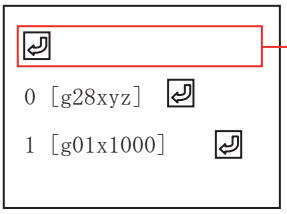
Precautions

(1) Always input registration number "n" at the head of a row.
(2) Only the character string enclosed in brackets [ ] is to be defined.
(3) If no configuration file is defined, no registration number is assigned to a user-defined key, or if a description is illegal, the default operation is performed as shown in the correspondence table above.

2.9.1 Format of Configuration File

This section describes the format of the configuration file (KEYDEF.TXT) using a type 1 description example.

(Configuration file description example)



Leaves a 1-row space.

(Supplement)

indicates a linefeed, and LF or CR LF is assumed to be a linefeed.

(1) Insert a row of only a linefeed at the top of KEYDEF.TXT.

(2) Describe a definition of each key in the second row and after. In this case,

- "0" means that character string "g28xyz" is defined as the operation of the "U1" key.
  As the actual operation, pressing the "U1" key inputs character string "G28 XYZ".
- "1" means that character string "g01x1000" is defined as the operation of the "U2" key.
  As the actual operation, pressing the "U2" key inputs character string "G01X1000".

Precautions

(1) When a row is prefixed by a number other than "0" or "7", it is assumed to be a comment line.

(2) The keys to be defined in the configuration file can be described in no particular order. If necessary, No. 7 can be first described.

(3) The range enclosed in brackets [ ] is assumed to be a definition. Characters following the right bracket "]" are assumed to be a comment.

(4) Only ASCII characters are available, excluding two-byte characters or Chinese characters.

(5) To describe a definition in the configuration file, be sure to leave a one-row space at the top. This is because the first row is not displayed by a parameter setting when the configuration file is opened on the edit screen.

(6) Describe only the keys to be defined.

(7) If you press another key while the defined characters are being output, it causes the defined-key output to be ended.

(8) When different defined-character strings are assigned to the same key in the file, the last assigned character string is input.

(9) When a key combined with the "ALTER" key is input (e.g. "ALTER + SHIFT + U1"), it functions as the user-defined key "SHIFT + U1", and the "ALTER" key is ignored.

(10) When a key with both the "CTRL" and "SHIFT " keys combined is specified (e.g. "CTRL + SHIFT + U1"), it functions as the user-defined key "SHIFT + U1", and the "CTRL" key is ignored.

(11) A character "A" to "B" at the head of a row can be defined in either uppercase or lowercase characters.

2.9.2 Definition Types of User-defined Keys

The user-defined keys are classified into two definition types: type 1 and type 2, which provide different operations as shown below.

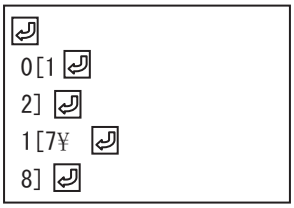
	Type 1	Type 2
Linefeed in brackets [ ]	Not handled as a semicolon ";".	Handled as a semicolon ";".
Dependence of the Caps Lock key	The defined characters are converted to uppercase or lowercase characters depending on the status of the Caps Lock key before they are input.	Characters are input as defined regardless of the status of the Caps Lock key.
Conversion of symbol characters	Symbol characters are converted to specific ones.	No symbol characters are converted.

2.9.2.1 Type 1

Linefeed in brackets [ ]

In the configuration file, a character string to be defined is described in the range enclosed in brackets [ ]. However, if a linefeed is included within the brackets, it causes a format error that assumes that no definition is described for that key.

(Example) Contents of KEYDEF.TXT



(Supplement)

- ↵ indicates a linefeed, and LF or CR LF is assumed to be a linefeed.

With the configuration file above registered, the following characters are input when the "U1" and "U2" keys are pressed.

Key input	Characters to be input	Supplement
U1	None	No characters are input because the format of the configuration file is invalid.
U2	78	If a linefeed is inserted in a row that includes symbol "\" at the end, it is assumed that the row is continued to the next file.

Handling the Caps Lock key

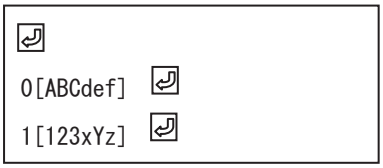
Characters are input in accordance with the status of the Caps Lock key.

Status of the Caps Lock key	Definition in uppercase characters	Definition in lowercase characters
ON	Lowercase characters are input.	Uppercase characters are input.
OFF	Uppercase characters are input.	Lowercase characters are input.

Note

(1) In NC, the Caps Lock key is set ON when the power is turned on.

(Example) Contents of KEYDEF.TXT



(Supplement)

- ↵ indicates a linefeed, and LF or CR LF is assumed to be a linefeed.

Input characters with the "U1" and "U2" keys input

Key input	At Caps Lock ON	At Caps Lock OFF
U1	abcDEF	ABCdef
U2	123XyZ	123xYz

Conversion of defined characters

There is a case in which the defined-characters described in the range enclosed in brackets [ ] are converted and input. When symbol "\" is described, the input characters (behavior) vary depending on the succeeding characters. The following shows the correspondence between the defined characters and input characters (behavior).

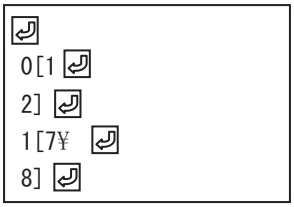
Defined character		Input character (behavior)	
(Alphabetical characters)		(The defined characters are input in uppercase or lowercase characters depending on the status of the Caps Lock key.)	
(Digits)		(Digits are input as defined.)	
\\		]	Right bracket
\		(If a linefeed is inserted in a row that includes symbol "\" at the end, it is assumed that the row is continued to the next file.)	
\t		The [→|] key (tab key) is input.)	
\n			(The [INPUT] key is input.)
\[		@	At mark
\]		[	Left bracket
\	Back slash	(No data is input. Symbol "\" other than the above is ignored.)	
!	Exclamation	!	Exclamation
"	Double quotation	*	Asterisk
#	Sharp	#	Sharp
$	Dollar	$	Dollar
%	Percent	%	Percent
&	Ampersand	'	Single quotation
'	Single quotation	:	Colon
(	Left parenthesis	)	Right parenthesis
)	Right parenthesis	(No data is input.)	
=	Equal	^	Hat
~	Tilde	(No data is input.)	
|	Pipeline	}	Right brace
`	Backquote	(No data is input.)	
{	Left brace	`	Backquote
}	Right brace	{	Left brace
+	Plus	~	Tilde
*	Asterisk	(	Left parenthesis
<	Left angle bracket	<	Left angle bracket
>	Right angle bracket	>	Right angle bracket
?	Question mark	?	Question mark
_	Underscore	=	Equal
-	Hyphen	-	Hyphen
^	Hat	&	Ampersand
@	At mark	"	Double quotation
[	Left bracket	(Not available)	
]	Right bracket	(Specified at the end of a definition.)	
;	Semicolon	;	Semicolon
:	Colon	+	Plus
,	Comma	,	Comma
.	Period	.	Period
/	Slash	/	Slash

2.9.2.2 Type 2

Linefeed in brackets [ ]

In the configuration file, a character string to be defined is described in the range enclosed in brackets [ ]. However, if a linefeed is included in this range, it is handled as a semicolon ";". This is assumed that a semicolon ";" is defined as one character for linefeed.

(Example) Contents of KEYDEF.TXT



(Supplement)

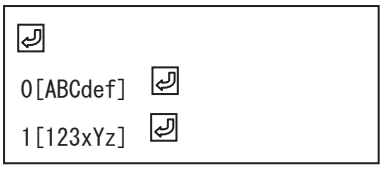
- ↵ indicates a linefeed, and LF or CR LF is assumed to be a linefeed.

Input characters with the "U1" and "U2" keys input

Key input	Input characters	Supplement
U1	1;2	A linefeed is handled as a semicolon ";".
U2	7;8	A linefeed is handled as a semicolon ";". This is handled as an independent symbol "\", and ignored.

Handling the Caps Lock key

Characters are input as defined regardless of the status of the Caps Lock key.



(Supplement)

- ↵ indicates a linefeed, and LF or CR LF is assumed to be a linefeed.

Input characters with the "U1" and "U2" keys input

Key input	Input characters
U1	ABCdef
U2	123xYz

Conversion of defined characters

The defined-characters described in the range enclosed in brackets [ ] are input as defined. However, when symbol "\" is described, the input characters (behavior) vary depending on the succeeding characters.
The following shows the correspondence between the defined characters and input characters (behavior).

Defined character		Input character (behavior)	
(Alphabetical characters)		(Characters are input as defined regardless of the status of the Caps Lock key.)	
(Digits)		(Digits are input as defined.)	
\\		\	Back slash
\t		The [→|] key (tab key) is input.)	
\n			(The [INPUT] key is input.)
\[		[	Left bracket
\]		]	Right bracket
\	Back slash	(No data is input. Symbol "\" other than the above is ignored.)	
!	Exclamation	!	Exclamation
"	Double quotation	"	Double quotation
#	Sharp	#	Sharp
$	Dollar	$	Dollar
%	Percent	%	Percent
&	Ampersand	&	Ampersand
'	Single quotation	'	Single quotation
(	Left parenthesis	(	Left parenthesis
)	Right parenthesis	)	Right parenthesis
=	Equal	=	Equal
~	Tilde	~	Tilde
|	Pipeline	|	Pipeline
`	Backquote	`	Backquote
{	Left brace	{	Left brace
}	Right brace	}	Right brace
+	Plus	+	Plus
*	Asterisk	*	Asterisk
<	Left angle bracket	<	Left angle bracket
>	Right angle bracket	>	Right angle bracket
?	Question mark	?	Question mark
_	Underscore	_	Underscore
-	Hyphen	-	Hyphen
^	Hat	^	Hat
@	At mark	@	At mark
[	Left bracket	(Not available)	
]	Right bracket	(Specified at the end of a definition.)	
;	Semicolon	;	Semicolon
:	Colon	:	Colon
,	Comma	,	Comma
.	Period	.	Period
/	Slash	/	Slash

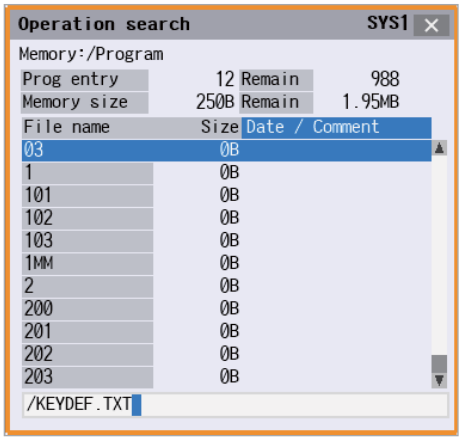
2.9.3 Editing a Configuration File

On the edit screen, you can edit the configuration file (KEYDEF.TXT) in "M01:\MSYS\CUSTOM". The edited data is enabled when the power is turned on again.

Edit – 

(1) Press the menu [Open]. → The list appears as a pop-up window.

(2) Input "KEYDEF.TXT". → "KEYDEF.TXT" is displayed in the input field of the list.



(3) Press the [INPUT] key. → The configuration file (KEYDEF.TXT) is displayed. Edit the file.

Note

(1) To input or output the configuration file, enter "/MSYS/CUSTOM" in the directory on [Mainte] - [I/O] screen.

Restrictions

(1) Up to eight key types can be registered.
(2) Up to 256 characters can be described in each definition. If this maximum number is exceeded, the definition is ignored.
(3) Only the keys on the operation board are available as the types of characters to be registered.
(4) When the edit screen is in the playback edit mode, the operation message "Can't execute playback edit" is displayed if an attempt is made to perform playback editing.
(5) When the configuration file (KEYDEF.TXT) is edited on the edit screen, ;(EOB) cannot be replaced.
(6) When the configuration file (KEYDEF.TXT) is edited on the edit screen, the first row is not read if the parameter "#19006 EOR disable" is set to "0". Therefore, if any characters are defined on the first row, their definition is deleted by an editing operation.
(7) The configuration file (KEYDEF.TXT) cannot be erased on the edit screen.
(8) Any user-defined keys are not displayed on the software keyboard.
(9) Always open the configuration file (KEYDEF.TXT) in regular editing mode regardless of the setting of the parameter "#8921 Mass Edit select". No editing can be carried out in mass-editing mode.
(10) On the edit screen, a block is split-displayed in words; however, spaces are not saved even if specified.
(11) On the edit screen, up to 255 characters can be input to one block, and the 256th character and after are cut to be displayed. If data is saved in this status, the 256th character and after will be cut.
(12) If the configuration file is not created, or if no defined character string is assigned to a key, no operation is performed when the "U1" to "U8" keys are pressed.

2.10 Outputting the Screen Information (Screenshot Capture Function) [M800VS/M80V Series]

When the parameter "#8121 Screen Capture" is set to "1" and then [SHIFT] key is kept pressing, the screen information displayed can be output to a USB memory or an SD card as a bitmap format file.

Operation method

(1) Set the parameter "#8121 Screen Capture" to "1".

(2) Display the screen to capture.

(3) Keep pressing the [SHIFT] key on the keyboard for about 3 seconds. → Machine beeps when the [SHIFT] key is pressed. Output of the bitmap file is completed when the machine beeps again after about 3 seconds.

(4) Reset the parameter "#8121 Screen Capture" to "0".

Bitmap file format

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

Number of colors	Color (256 colors)
Size	308,278 bytes (640 × 480)
File name	A day of the week_Month_Day_Hour_Minute_Second_Year.bmp (Example) When the file is output on "Wednesday, Jan. 23, 2019 at 12:18:38" WED_JAN_23_12_18_38_2019.bmp

Precautions

(1) Still images can be output with this function.

(2) The bitmap file cannot be output during cycle start.

(3) Do not output the bitmap file while any axis is moving by the manual operation mode etc.

(4) The screen operation cannot be performed while the bitmap file is being output to a USB memory or an SD card. Do not turn the power OFF while a file is being output. The USB memory or SD 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 can be created, however; it is not correctly displayed.

3

Inputting a Machining Program and Defining Variables

This chapter explains creating and editing the machining program. Generally, operations are performed on the edit (Edit) screen.
Refer to "Programming Manual" for details on G code programming.

3.1 Inputting a Machining Program

When using the program created by external device, input it to NC by using program input/output function. For information on the data input/output function, refer to "5.1 Inputting and Outputting a Program".

3.2 Creating and Editing a Machining Program



Program editing (add, erase, change) can be performed in the NC memory, NC memory 2, HD, memory card (front-side SD card), DS (SD card inside the control unit) or USB memory.

This function is used for machining program, MDI program and fixed cycle program.

On the edit screen, editing (add, erase, change or interactive cycle insertion) a machining program and cursor position search can be performed.
There are 4 types of display as shown below:

Multi-program display type	Enables you to edit two programs using each editing area.
G-code guidance display type	Enables you to edit a program referring to the G code guidance.
Playback display type	Enables you to edit a program trying sample machining.
Finish shape display type	Enables you to edit a program confirming the finished shape.

Selected display type is retained after screen transition.

The tab under the editing area is switched depending on the selected display type.
Some tabs may not be displayed depending on the specifications and parameters.



For details on switching the display type, refer to "3.6.1 Changing the Display".

You can edit the program on the monitor screen as well. For details, refer to "10.9 Correcting the Machining Program".

Note

(1) For 3-program/4-program simultaneous editing display on 15-type/19-type display unit, tabs are not displayed. For details on switching the screens, refer to "3.6.1 Changing the Display".

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:

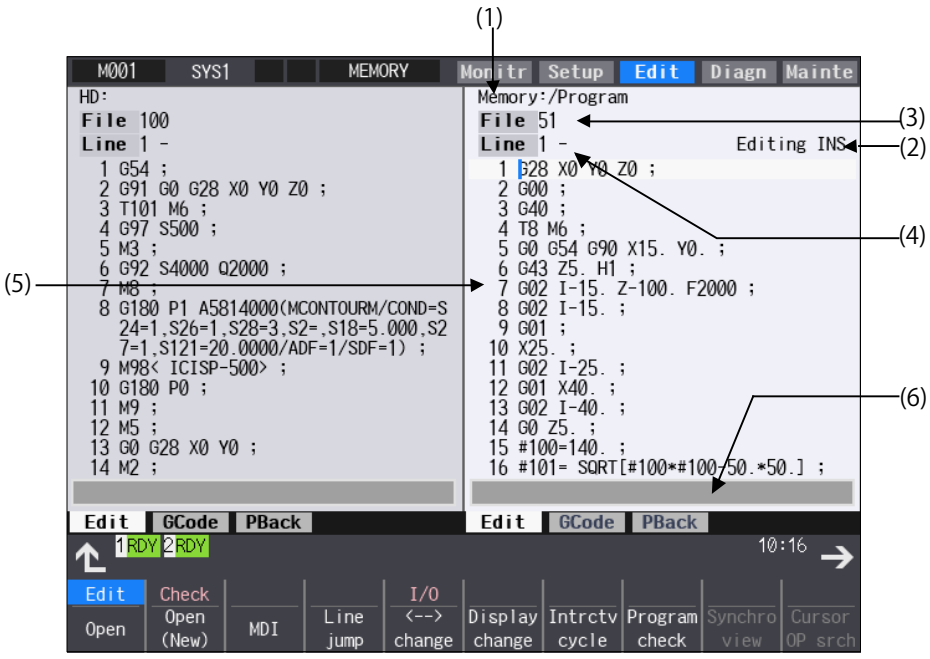
Mass-editing is not available for programs stored in NC memory or NC memory 2.

Model	Conditions of mass-editing	Maximum edit size
M800V Series M80VW Series	♦The storage location of the machining program to be edited is either an HD, a USB memory, a memory card or a DS. ♦The size of the file is 1.0 MB or more. (Except when the parameter "#8910 Edit undo" is "0", which needs 2.0 MB or more.)	1 GB
M80V Series	♦The storage location of the machining program to be edited is either a memory card, a USB memory or a DS. ♦The size of the file is 0.5 MB or more.	

⚠ CAUTION

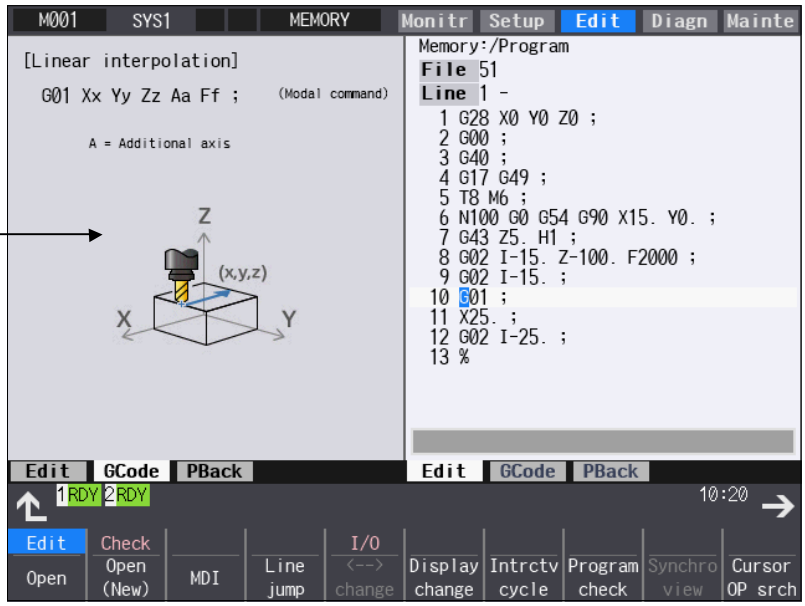
The program including a character of any language other than the display language is not correctly displayed. Do not edit such a program. Any part of the program other than the comment part may also be changed if edited.

■Multi-program display type



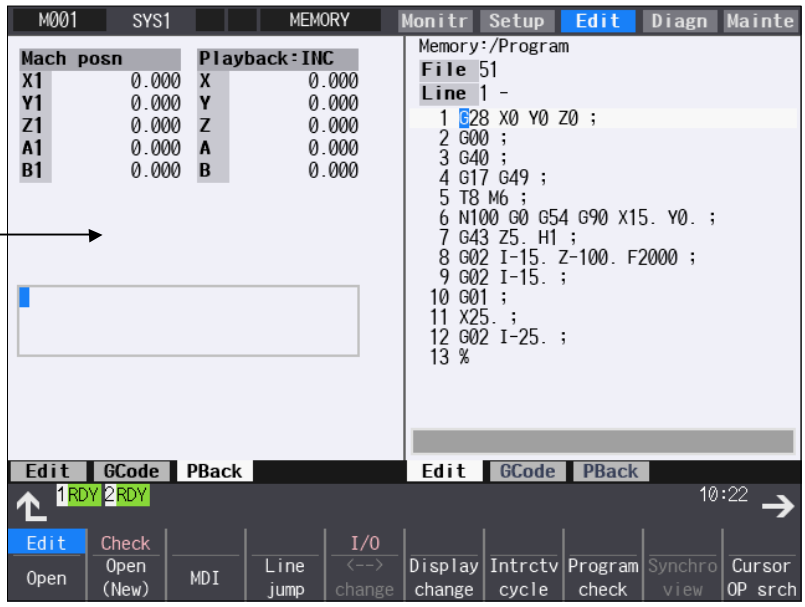
■G-code guidance display type

(7)

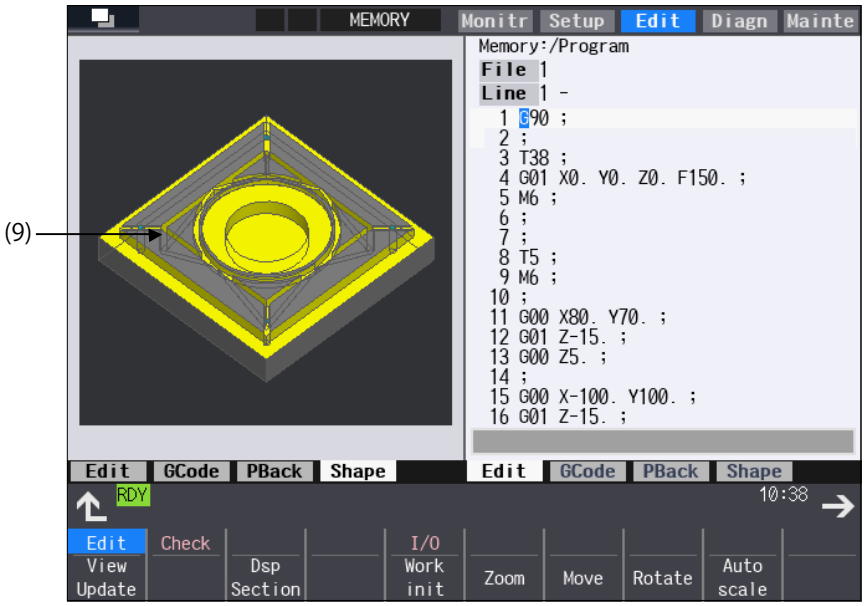


■Playback display type

(8)



■Finish shape display type



Display items

Display items		Details
(1)	Path display	The path of the currently opened program file is displayed. (Example) Memory:/program When the path is too long, the portion exceeding the display area is not displayed.
(2)	N No. incremental value display/Edit type display/Current edit display "Editing"/Insert mode display "INS"	<N No. incremental 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"> The overwrite mode and insert mode are switched by pressing the [INSERT] key. This is displayed when the insert mode is selected.
(3)	Program name display	This displays the file name of the program currently being edited. If the file name exceeds the display area, the last character is "*". This displays "MDI" when the MDI program name is being edited.
(4)	Top line of the displayed program/Display position	This displays the top line of the displayed program. During mass-editing, the position of the currently displayed page is displayed in percentage (rate) when the entire size of the program is assumed to be 100%.
(5)	Line number/Program display	<Line number> This displays the last 3 digits of the program line number. When over two or more lines are displayed on the screen because one line is long, the line No. does not attach to the beginning of the next line. 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 that the cursor is on is highlighted. The line is not emphasized during mass-editing.
(6)	Input section	This inputs the program line number and search character string.
(7)	G code guidance display	This displays G code guidance. For details, refer to "2.5.3 G Code Guidance".
(8)	Playback display	This displays playback status. For details, refer to "3.7.5 Playback Editing".

Display items	Details
(9) Finished shape display	This displays finished shape status of a machining program. For details, refer to "3.7.7 Displaying the Finished Shape".

Menus

Menus	Details
Open	This edits and references the existing programs. When the program number is designated and the [INPUT] key is pressed, the program contents appear and can be edited.
Open (New)	This creates a new program. When the program number is set and the [INPUT] key is pressed, new comment or program contents can be created.
MDI	This edits the MDI program. When this menu key is pressed, the MDI program appears and can be edited.
Line jump	When the line number is set and the [INPUT] key is pressed, the program data for the set number and later is displayed on the screen. The cursor moves to the set line number.
% 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] 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.
<--> change	The area that can be edited is switched between the left and right areas. This menu is available for multi-program display type or the multi-part system simultaneous program editing (3-program editing).
Display change	The display type (multi-program display, G code guidance display, playback display or finish shape display) is switched.
Intrctv cycle	This edits the cycle interactively. <Note> ♦If the parameter "#8991 Interactive cycle" is "0", this menu is grayed out and non-selectable. ♦If there is no specification of the interactive cycle insertion, this menu is not displayed.
Program check	Program check is carried out for the program displayed in the area that can be edited.
Synchro view	A blank line is inserted so that the timing synchronization symbols (the timing synchronization operations of the machining program) are aligned horizontally. <Note> ♦This menu is grayed out and non-selectable in the following cases. - When the parameter "#1285 ext21/bit0" (Multi-part system program management) is set to "0" (invalid). - When two or more edit areas do not display the same name programs of different part systems that are stored on NC memory or NC memory 2. - When the program numbers displayed in the left and right editing areas are different.
Cursor OP srch	An operation search is carried out for the currently opened program with the block located by the cursor at the top. When the operation search is completed, the display switches to the monitor screen. <Note> ♦If the program is being edited, this menu is grayed out and non-selectable. ♦If the displayed part system is a sub-part system which does not accept the operation search, this menu is grayed out and non-selectable.
String search	When the character string is designated and the [INPUT] key is pressed, that character string is searched for.
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] key is pressed, the character string is replaced. This menu cannot be selected during mass-editing.

Menus	Details
Line copy	This copies the designated line (multiple lines can be copied.) This menu cannot be selected during mass-editing.
Char copy	This copies the character string designated by the cursor (multiple lines can be copied.)
Paste	This inserts the most recently copied contents to the line before the cursor or the position before the cursor.
Line clear	This erases the designated line (multiple lines can be erased). This menu cannot be selected during mass-editing.
Program search	Pressing this menu switches to operation search mode. <Note> ♦This menu is displayed when the parameter "#1759 cfgPR09/bit2" (Operation search from edit screen) is set to "1". ♦This menu is grayed out and non-selectable in the following cases: - When a program cannot be edited due to edit lock B, C, program display lock, or data protection. - While a program is being edited.
<--> change	The area that can be edited is switched between the left and right areas. This menu is available for multi-program display type or the multi-part system simultaneous program editing (3-program editing). <Note> ♦This menu is displayed when the parameter "#1759 cfgPR09/bit4" (Edit area change menu additional display) is set to "1". ♦This menu and the [Program search] menu cannot be used simultaneously. When the parameter "#1759 cfgPR09/bit2" (Operation search from Edit screen) is set to "1", the [Program search] menu is displayed instead of this menu.
Miss warning	This enables the input mistake check for the program being edited. This menu cannot be selected during mass-editing.
Next miss	This moves the cursor to the next input mistake warning location. This menu cannot be selected during mass-editing.
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. <Note> ♦When the parameter "#8939 Undo confirm msg" is set to "1", the menu is highlighted while the operation message is displayed.
N auto add	The sequence No. can be added automatically by pressing the [INPUT] key after designating an additional value. This menu cannot be selected during mass-editing.
MDI regist	This registers the MDI program in the memory. This can be performed only when the MDI program is displayed. This menu cannot be selected during mass-editing.
Vibe cut	This assists in adjusting the machining conditions for vibration cutting control. <Note> ♦This menu is displayed when the vibration cutting expansion unit (FCU8-EX744) is mounted. ♦This menu is grayed out and non-selectable when no program is displayed.
Erase file	This deletes the programs. When the name of the program to be erased is designated, the designated file is deleted.
Display setting	This menu switches the display format of multi-part system simultaneous program editing (2-program/3-program or 2-program/4-program editing). <Note> ♦This menu is available for 15-type or 19-type.

Menus	Details
PSHAPE	This menu switches to the display of machining shape appropriate to the command which is displayed at the time of machining program input. <Note> ♦This menu is displayed only when the optional specification of finish shape view programming is enabled. ♦If the "Shape" tab is not displayed, this menu is grayed out and non-selectable.
Guide M/L	This menu switches G code guidance display (M/L system). The switchable G code system follows the parameter "#11082 Lsys_change_Gtype". <Note> ♦This is displayed only when the program format switch specification is valid and the parameter "#1254 set26/bit1" is set to "1". ♦When the G code guidance is not displayed, this menu is grayed out and non-selectable.
Pback next ax	The axis to be displayed in the playback editing is switched from the first to fifth axes and among the sixth and further axes. <Note> ♦This menu is enabled if the playback editing (L system) is enabled and the number of axes is 6 or more.
File close	This closes the pop-up window and quits this function.
Upward Tsearch	These menus search for T code upward or downward.
Down Tsearch	
Program Convert	This function converts two or more rotary axis commands to one rotary axis command, in a block of the machining program where the tool center point control is commanded. <Note> ♦This menu is not displayed in the following cases. - The optional specification of tool center point control is disabled. - The parameter "#1285 ext21/bit0" (Multi-part system program management) is "1" (Valid).

Sub-menus of [Open]/[Open (New)]/[Erase file] menus

Menus	Details
Memory	This selects NC memory.
Memory2	This selects NC memory 2.
HD	This selects the hard disk. This menu is not displayed on M800VS/M80V Series.
Memory card	This selects the front-side SD card.
DS	This selects the built-in SD card.
USB Memory	This selects USB memory.
Top jump	This moves the cursor to the top line.
Bottom jump	This moves the cursor to the last line.

Menus	Details
Close	This closes the pop-up window and return to the edit screen.
Comment nondisp	This changes whether to show or hide the comment field. When the comment is hid, the menu is highlighted.
Delete image	This erases the image data associated with the selected machining program. <Note> ♦This menu is displayed only when the menu [Erase file] is selected.
Preview	This switches to show or hide the preview (detailed information).
Change Disp	This changes the display format of the file/directory list.
List update	This updates the list contents. (The latest contents of the currently selected device and directory are listed.)
Sort change	This changes the method that the list is sorted.

Sub-menus of [Displaysetting] (For 15-type/19-type)

Menus	Details
2 edit	This switches the display mode to the program simultaneous program edit (2 programs).
3 edit	This switches the display mode to the program simultaneous program edit (3 programs or 4 programs). <Note> ♦You can select either 3 programs or 4 programs with the parameter "#8999 Simul edit - View".
4 edit	

Sub menus of [String search]

When the parameter "#19710 Edit-Upward search" is set to "1", the sub menus are displayed.

Menus	Details
Upward search	This switches the search direction upward.
Down search	This switches the search direction downward.
Quit search	This finishes the string search.

Operation method (Creating a new machining program)

(1) Press the menu [Open (New)] on the edit screen.

(2) Select the device.
(Example) Menu [Memory]
→ The selected device name and directory (memory: /program) appear in the device name and directory display columns.
For the devices other than NC memory or NC memory 2, root directory is selected.
The device on the side being edited is displayed first. If a file is not opened, NC memory will be opened.

(3) (For the devices other than NC memory or NC memory 2)
Press [↑], [↓], PAGE, or PAGE key to move the cursor to the directory with the machining program or the directory to be newly created.
If the list contents differ from the actual device or directory, press the menu [List Update].

(4) (For the devices other than NC memory or NC memory 2)
Press the [INPUT] key.
→ Moves into the directory.

(5) Enter the program file name to be newly created.

(6) Press the [INPUT] key.
→ When new creation is enabled, an EOR-only program is created.
The list display is closed.

(7) Edit the machining program.
→ Refer to "3.6 Editing Operations".

(8) Press the [INPUT] key.
→ The edited machining program is saved to the device.

Precautions for creating a new machining program

(1) If an existing program number is designated, it causes an error.

(2) The part enclosed in parentheses ( ) in the head block of the program is used as a comment.

(3) If an existing file is set, the message "Designated file already exists" appears.

(4) Single-byte digits, single-byte uppercase alphabet characters, and Windows-recognizable single-byte symbols are available for the file name or directory name. Therefore, a double-byte code file name such as a Japanese file name cannot be used.
Characters which cannot be used: \ / : , * ?“ < > | a to z and space
Also, the followings cannot be treated as file names:

- File name with extension "$$$", "$$0", "$$1", "$$2", "$$3", "$$4", "$$5", "$$6", "$$7", "$$8", or "$$9"
- "0" (File named "single-byte zero")

(5) A program, which has a file name consisting of 33 or more characters, cannot be newly created.

(6) If the parameter "#8936 Delete leading 0" is set to "1" and the file name is designated only by digits, the existing file, edit lock B, and edit lock C are checked by deleting "0" at the head.
For details, refer to "5.8.6 Leading Zero".

Operation method (editing machining program)

(1) Press the menu [Open] on the edit screen.

(2) Select the device.
(Example) Menu [Memory]
→ The selected device name and directory (memory: /program) appear in the device name and directory display columns.
For the devices other than NC memory or NC memory 2, root directory is selected.
The device on the side being edited is displayed first. If a file is not opened, NC memory will be opened.

(3) (For the devices other than NC memory or NC memory 2)
Press [↑], [↓], PAGE, or PAGE key to move the cursor to the directory with the machining program.
If the list contents differ from the actual device or directory, press the menu [List Update].

(4) (For the devices other than NC memory or NC memory 2)
Press the [INPUT] key.
→ Moves into the directory.

(5) Press [↑], [↓], PAGE, or PAGE key to move the cursor to the machining program to be edited.
The name of the machining program to be edited can be input in the input area.

(6) Press the [INPUT] key.
If the program name has been entered in the input area, the program contents is displayed.
→ The program appears at the top if the file can be opened.
The cursor moves to the head character of the program.
The mode changes to the overwrite mode.
(When it is the initial insert mode, "INS" is displayed to the right of the file name and the mode changes to the insert mode.)
The list display is closed.

(7) Edit the machining program.
→ Refer to "3.6 Editing Operations".

(8) Press the [INPUT] key.
→ The edited machining program is saved to the device.

Precautions for editing machining program

(1) An error occurs if a nonexistent program No. is set.

(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 [INPUT] key is pressed.

(3) The fixed cycle program list is displayed when the parameter "#1166 fixpro" is set to "1".

(4) Editable size differs depending on each device. When the editable size is exceeded, the operation message "Can't edit because of size over" appears.

(5) "Loading" is displayed in flickering until the file is opened.

(6) Editing is not possible for a program with 33 or more file name characters.

(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 -> Reset & Rewind -> Edit: The program top is searched again.

(8) If an attempt was made to open the currently edited file on the custom screen from this screen, the operation message "Read-only as the file is being edited" may appear.

(9) If an attempt was made to edit the currently edited file in the custom screen from this screen, the operation message "Read-only file - Can't edit" appears and may not be able to edit.

(10) If the protection level ("Program edit" data output) designated on the protection setting screen is higher than the current operation level, the machining program cannot be opened (message "Data protect" is displayed). However, if the protection level is set to a higher value on the protection setting screen while a machining program remains opened, the machining program is kept opened even after the display has returned to the edit screen. To hide the machining program after changing the protection level, press the menu [Close] to close the machining program, and change the protection level on the protection setting screen. For details, refer to "2.8 Changing the Operation Level (Protect Setting Screen)".

(11) Display will be changed as follows depending on the value set in the user parameter "#19006 EOR Disable".

<When the parameter "#19006 EOR Disable" = "0">

- When a file including "%" is opened in the edit screen, characters up to "%" will be displayed. By saving the file, characters after "%" will be discarded.
- When a file not including "%" is opened, "%" will be displayed at the end of the edit screen. By saving the file, "%" will also be saved.
- When an external device file, other than NC memory or NC memory 2, is opened, it will be displayed in the edit file with skipping the first line.

<When the parameter "#19006 EOR Disable" = "1">

- The contents of a file is displayed on the edit screen as it is.
- When a file whose last character is other than "%" is opened, "%" will be displayed at the end of the edit screen. "%" will be saved by saving the file. For example, if the end of the file is "%" + "CRLF", an extra "%" will be added at the end.
- When an external device file, other than NC memory or NC memory 2, is opened, the first line will not be skipped.
- "Line copy" and "Line clear" can be made at the lines other than the last line which includes "%".

Opening a machining program by specifying a sequence No.

(1) Press the menu [Open] on the edit screen.

(2) Select the device.
(Example) Menu [Memory]

→ The selected device name and directory (memory: /program) appear in the device name and directory display columns.
The menu [Memory] is highlighted.
For the devices other than NC memory or NC memory 2, root directory is selected.
The device on the side being edited is displayed first. If a file is not opened, NC memory will be opened.

(3) Input "(Machining program name)/(Sequence No.)" in the input section.
(Example) When specifying sequence No. 100 of program 1
1/100

(4) Press the [INPUT] key.

→ The program appears from the specified sequence No. if the file can be opened.
The cursor moves to the head character of the sequence No. and the mode changes to the overwrite mode.
(When it is the initial insert mode, "INS" is displayed to the right of the file name and the mode changes to the insert mode.)
The list display is closed.

Note

(1) When the file does not exist, the message "Designated file does not exist" appears and the file is not opened.

(2) When the nonexistent sequence No. or the value which cannot be treated as sequence No. is specified, the message "Nonexistent sequence #: display from top" appears and the head of the program is displayed.
Numbers 1 to 99999999 can be treated as sequence No.

(3) "Loading" is displayed until the file is opened. The message flashes every second.

3.3 Registering and Editing the MDI Program

Edit – 

Operation method for editing MDI program

(1) Press the menu [MDI] on the edit screen. → The menu is highlighted.
The MDI program is displayed from the head of the editing side.The cursor moves to the head character of the program.
The cursor moves to the head character of the program.
The mode changes to the overwrite mode.
(When it is the initial insert mode, "INS" is displayed to the right of the file name and the mode changes to the insert mode.)

(2) Edit the MDI program. → Refer to "3.6 Editing Operations".

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 [INPUT] key is pressed, the head block will be searched, and the message "MDI Set ended" 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 [INPUT] key is pressed.

(2) If the [INPUT] 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.
Note that this function cannot be used with NC memory 2.

(3) When the parameter "#1288 ext24/bit0" (MDI program clear) is set to "1", the MDI programs are cleared when MDI operation ends, the power is turned ON again, reset is input and emergency stop is canceled, and only % programs are saved.

- When the parameter "#1285 ext21/bit0" (Multi-part system program management) is set to "1" (valid), the MDI program of the part system in which MDI operation ends or the reset signal issues is cleared.
- The MDI programs before editing is cleared during the MDI program editing.
- Reset input refers to NC reset 1, NC reset 2 and reset & rewind.
- MDI program clear is performed regardless of operation mode.
- When MDI program is cleared during the character string replacement confirmation waiting or line clear confirmation waiting, the character string replacement or line clear is canceled and each menu highlight returns to normal.
- When MDI program is cleared after the range of the target block for line copy and line clear has been set, the designated machining program is de-highlighted.

Operation method for registering MDI program

(1) Press the menu [MDI regist] on the edit screen.

(2) Select the program to register, and then press the [INPUT] key.

Note

(1) The program is registered as a general program of NC memory.
When the fixed cycle or MTB macro is selected with the parameter "#1166 fixpro" (Fixed cycleediting), the program cannot be registered. The "Save not possible" message appears.

(2) Program cannot be registered while "Editing" is being displayed. The "Save not possible" message appears. Press the [INPUT] key to be "MDI Set ended".

(3) When the general program remaining capacity is less than the value of MDI program to be registered, the program cannot be registered. The "Memory over" message appears.

(4) When the number of remaining general programs is zero, the program cannot be registered. The "Program entry over" message appears.

(5) When the [INPUT] key is pressed without entering the program name, "Setting error" occurs.

(6) If the program name is input with only numbers when the parameter "#8936 Delete leading 0" is set to "1", the number without the leading zero is registered as the program name. However, when a number without a leading zero is outside the range 1 to 99999999, the leading zero is not deleted.

(7) If the input program name already exists in the NC memory, the MDI program cannot be registered during the operation, program restart or program checking. The operation message "Executing automatic operation", "Program restarting" or "Program checking" appears.
If the program name is input with only numbers when the parameter "#8936 Delete leading 0" is set to "1", the program is checked with the leading zero deleted. (Example: If the MDI registration is specified as "0001" when the program "1" is in automatic operation, the "Executing automatic operation" message appears and the registration cannot be performed.)

(8) When the input program name already exists in the NC memory and it is not being in operation, the "Overwrite this file?(Y/N)" message appears. When the [Y] or [INPUT] key is input, the program name is overwritten, and when a key other than those is pressed, the registration is canceled.
If the program name is input with only numbers when the parameter "#8936 Delete leading 0" is set to "1", the program is checked with the leading zero deleted. (Example: If the MDI registration is specified as "0002" when the program "2" already exists in the NC memory and it is not in the operation, the "Overwrite this file?(Y/N)" message appears.)

(9) When the program name to be input is subject to edit lock (edit lock B: 8000 to 9999, edit lock C: 9000 to 9999), the message "Edit lock B" or "Edit lock C" appears, and the program name cannot be registered.
If the program name is input with only numbers when the parameter "#8936 Delete leading 0" is set to "1", the program is checked with the leading zero deleted. (Example: If the MDI registration is specified as "008000" when the edit lock B is valid, the "Edit lock B" message appears and the program cannot be registered.)

(10) When the data protection is valid, "Data protect" is displayed and the program cannot be registered in the NC memory.

3.4 Registering and 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 MTB.

Note

(1) Since the G.1 command is used to perform the movement of a fixed cycle program, do not edit the contents of the program being described before the G.1 command. Do not delete the contents or do not insert any other commands than the G.1 command. If you edit the program in such a manner, the fixed cycle program is not performed properly.

(2) When a fixed cycle program is executed, the modal information of the dedicated group 01 is used. This intends that the modal of the main program, which calls the fixed cycle program, is not influenced by the fixed cycle program. Therefore, you can edit the fixed cycle program without considering the modal of the main program. In the fixed cycle program, the modal of the group 01 after the G.1 command is "G00".

Refer to "Programming Manual" for details on the fixed cycle program.

3.4.1 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 the fixed cycle program operation, return the parameter to "0".

Note

(1) Parameter "#1166 fixpro" will be set to "0" when the power is turned OFF.

3.4.2 Transmitting and Erasing the Fixed Cycle Program

Transmit/erase the fixed cycle program from the Data I/O screen. Check that fixed the parameter "#1166 fixpro" is valid. The operating procedure is the same as a user machining program.

3.5 Erasing a File

Edit – 

Operation method (Erasing a file)

(1) Press the menu [Erase file] on the edit screen.

(2) Select the device.
(Example) Menu [Memory]
→ The selected device name and directory (memory: /program) appear in the device name and directory display columns.
For devices other than NC memory or NC memory 2, the root directory is selected.
The device on the side being edited is displayed first.
If a file is not opened, NC memory will be opened.

(3) (For devices other than NC memory or NC memory 2)
Press [↑], [↓], PAGE, or PAGE key to move the cursor to the directory with the machining program.
If the list contents differ from the actual device or directory, press [List update].

(4) (For devices other than NC memory or NC memory 2)
Press the [INPUT] key.
→ Moves into the directory.

(5) Press the [↑], [↓], PAGE or PAGE key to move the cursor to the program to be searched.
The name of the machining program to be erased can be input in the input area.

(6) Press the [INPUT] key.
→ The operation message "Erase?(Y/N)" is displayed.

(7) Press the [Y] or [INPUT] key.
Press the [N] key to cancel.

Note

(1) The file is not erased in the following cases.
- The file that is to be erased is currently being used in automatic operation.
- The file that is to be erased is subject to edit lock B or C.
- Data protection is valid.
- The file to be erased is in the "Program restarting" state.

(2) The image data associated with the file will also be erased.

Operation method (Erasing image data)

This will erase the image data associated with a machining program.

For details on how to link an image to a program, refer to "4.1.14 Associating Graphic Check Images with a Machining Program".

(1) Perform steps (1) to (4) from "Operation method (Erasing a File)".

(2) Use the [↑], [↓], PAGE, or PAGE key to move the cursor to the machining program to be edited.
The name of the machining program associated with the imaged data can be input in the input area.

(3) Press [Delete image]. → The operation message "Erase?(Y/N)" is displayed.

(4) Press the [Y] or [INPUT] key.
Press the [N] key to cancel. → The image data associated with the machining program will be erased.
List view will be updated.

3.6 Editing Operations

Edit – 

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 [INPUT] 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 this and the following sections.
These editing operations are common for the machining program and MDI program.

Note

(1) The initial mode (overwrite/insert) at start of editing and the [DELETE] key operation (DELETE/BackSpace) can be selected with the parameters. The operation with overwrite mode and DELETE operation is explained in this manual unless otherwise specified.

⚠ 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.

3.6.1 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.

Switching the display type

Press the menu [Displaychange] to switch the display type.

The same operation can be performed with the → key.

The display is switched in the order of multi-program display type -> G code guidance display type -> playback display type -> finished shape display type -> multi-program display type...

Note

(1) Press ← to switch in the inverse order.

(2) Some tabs may not be displayed depending on the specifications. When only the edit tab is displayed, the menu [Displaychange] cannot be selected.

Changing the display using the page up/down keys (PAGE : Previous page, PAGE : Next page)

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.

PAGE key: This displays one page of following lines of the current bottom line.
The cursor moves to the top line of the screen.

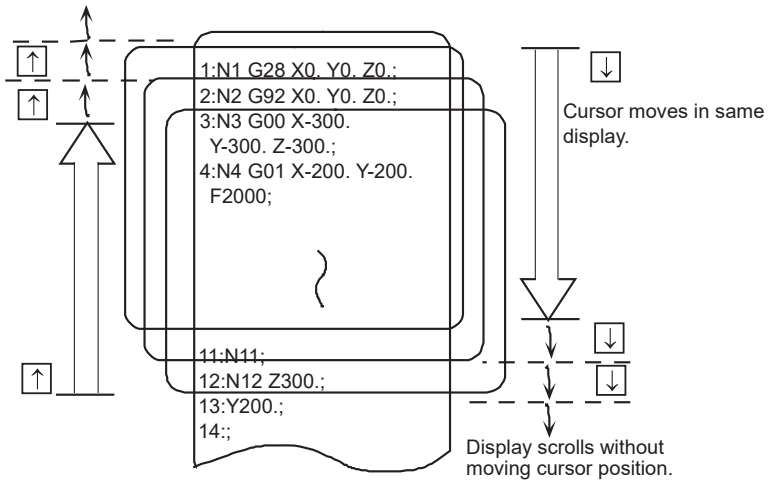
Note

(1) The 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 program of one line number extends over multiple lines>

■For the regular editing

When the parameter "#8990 Edit-up/down keys" is set to "0":
Press [↑], [↓] keys to move the cursor back and forth on the line number basis.

When the parameter "#8990 Edit-up/down keys" is set to "1":
Press [↑], [↓] keys to move the cursor to the previous or next line.

■For mass-editing

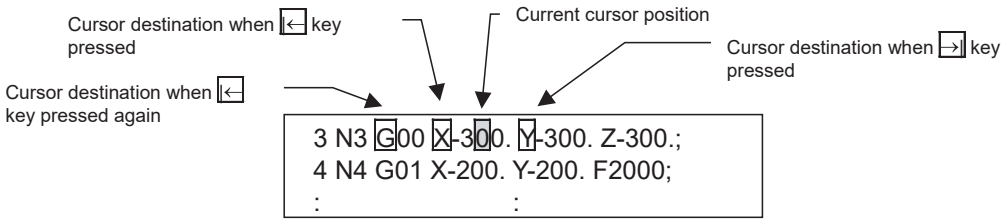
Press [↑], [↓] keys to move the cursor to the previous or next line.

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

(1) Pressing [|←] [→|] keys or [←] [→] keys does not make any difference in mass-editing.

Split display by word unit

The program is split by word unit and displayed.

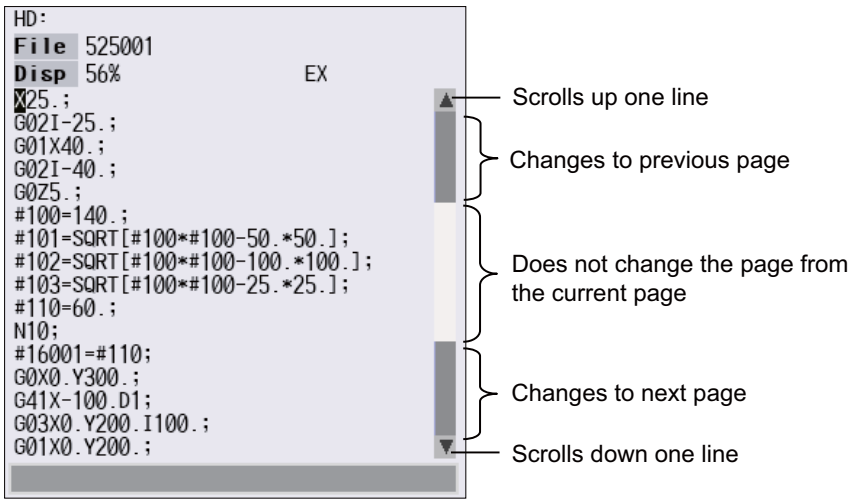
In mass-editing, program is not split by word unit and displayed as text data.

Switching the page by the scroll bar (Mass-editing)

For mass-editing, if you touch the scroll bar, the page is switched or the screen is scrolled by one line.

If you touch [▲] or [▼] of the scroll bar, the screen is scrolled up or down by one line. Also, if you touch the empty portion of the scroll bar, the page is switched to the next page or previous page.

Pages cannot be changed by dragging the slider.



3.6.2 Displaying an Arbitrary Line

Operation method

(1) Press the menu [Line jump].

(2) Input the line No.
(Example) 6 [INPUT]
→ The program appears with the designated line No. at the head.

Note

(1) When "0" is input, the cursor moves to the first line.
(2) When "E" is input, the cursor moves to the end.

3.6.3 Rewriting Data

Operation method (Initial data overwrite mode)

(1) Move the cursor to the position where data is to be inserted.
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 the 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, the cursor does not move.

Note

(1) 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 "3.6.5 Overwriting Data (Initial Data Insertion Mode)".)

Operation method (Initial data insertion mode)

(1) Move the cursor to the position where data is to be inserted.
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. The data from the cursor position moves to the right.
The cursor moves one space (character) at a time to the right as the data is set.

(3) 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 message "Editing" disappears.

Precautions

(1) When attempting to switch to another function such as the program check, or to another screen such as the monitor screen when "Editing" displays, a message checking whether or not to save changes appears. 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.)

(2) When attempting to switch to another function such as the program check, or to another screen such as the monitor screen when "Editing" displays, a message checking whether or not to save changes appears. If the [Y] key is pressed and changes still cannot be saved due to insufficient memory or other reasons, the message "Memory over" appears and the screen will not be changed. The screen displayed next will be as follows, depending on which of operations (a) or (b) is subsequently performed.
The same will occur if [INPUT] is pressed before the screen switch operation and changes cannot be saved due to insufficient memory or other reason.

(a) When changing to another function or to another screen such as the monitor screen without editing, the screen will switch without displaying the operation message. If the display returns to the program edit screen again, the previously saved contents are displayed.

(b) When changing to another function, or to another screen such as the monitor screen after editing (data setting key operation, data correction key operation, or the [Paste], [Line clear] or [String replace] operations), a message appears checking whether or not to save changes.

(3) When editing a program that is being checked, the "Checking other programs. Reset?(Y/N)" message is displayed. Press the [Y] or [INPUT] key to reset the check, and the edit operation can be continued. When pressing any other key, "Program checking" displays and the program cannot be edited.

(4) Up to 255 characters can be input in one block in regular editing and 127 characters in mass-editing.

(5) "Saving" is displayed in flickering during saving.

3.6.4 Inserting Data (Initial Data Overwrite Mode)

Operation method

(1) Move the cursor to the position where data is to be inserted.

(2) Press the [INSERT] key. → The mode changes to the insertion mode. "INS" appears to the right of the file name. The cursor changes to '|'.

(3) Set the data.

(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.

(2) The insertion mode will be finished if a key such as [DELETE], [C.B], [CAN], [INPUT], [↑], [↓], [←], [→], [|←], [→|], PAGE or PAGE pressed.

(3) For normal editing, EOB(;) is added to a row without EOB(;), and the cursor moves to the next line. A row with EOB(;) is skipped, and the cursor moves to the next line. (No cursor movement.)

(4) During mass-editing, linefeed is inserted every time EOB (;) is input. (Cursor moves to the head of the next line.)

3.6.5 Overwriting Data (Initial Data Insertion Mode)

Operation method

(1) Move the cursor to the position where data is to be inserted.

(2) Press the [INSERT] key. → The mode changes to the overwrite mode. "INS" appears to the right of the file name. The cursor changes to '■'.

(3) Set the data. → 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.

(4) When the setting is completed, press the [INPUT] key. → The set data is fixed. The mode returns to the insertion mode, "INS" is displayed in the right side of the file name, and the message "Editing" disappears. The cursor returns to '|'.

Note

(1) The insertion mode will be finished if a key such as [DELETE], [C.B], [CAN], [INPUT], [↑], [↓], [←], [→], [|←], [→|], PAGE or PAGE pressed.

(2) For normal editing, EOB(;) is added to a row without EOB(;), and the cursor moves to the next line. A row with EOB(;) is skipped, and the cursor moves to the next line. (No cursor movement.)

(3) During mass-editing, linefeed is inserted every time EOB (;) is input. (Cursor moves to the head of the next line.)

3.6.6 Deleting Data

Operation method (Deleting one character) (Deleting by DELETE key action ("#8967 DELETE key action" is set to "0"))

(1) Move the cursor to the character to be deleted.

(2) Press the [DELETE] 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 [INPUT] key. → The edited machining program is saved to the device and "Editing" disappears.

Operation method (Deleting one character) (Deleting by BackSpace key action ("#8967 DELETE key action" is set to "1"))

(1) Move the cursor to the right side of the character to be deleted.

(2) Press the [DELETE] key. → 1 character at the left side of the cursor position is deleted, 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 [INPUT] 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 erased.

(2) Press the [C.B] 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 [INPUT] 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.
(2) Last line only with "%" cannot be deleted.
(3) "Executing" is displayed during line clear. The message flashes every second.

Operation method (Deleting designated lines)

(1) Press the menu [Line clear].

(2) Designate the deletion range. → The background color for the range to be deleted turns light blue.

(Example) From line 8 to line 10
8/10 [INPUT]

When deleting 1 line, it is also possible to make a selection using the [↑], [↓] keys and designate by pressing the [INPUT] key.

(3) Press the [Y] or [INPUT] key.
Press the [N] key not to delete.

Note

(1) If the [INPUT] key is pressed without setting a line No., the line where the cursor is will be deleted.
(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
(3) "Executing" is displayed in flickering during line clear.
(4) During mass-editing, a line to be deleted cannot be designated.

Operation method (Deleting several lines (one screen of data) displayed on a screen)

(1) Press the [CAN] key. → The editing data (machining program) for one screen is deleted and "Editing" appears.

(2) Press the [INPUT] key. → The edited machining program is saved to the device and "Editing" disappears.

3.6.7 Searching for Character Strings

The specified character string is searched downward from the character after the cursor position.

When the parameter "#19710 Edit-Upward search" is set to "1", the search can be performed upward.

Operation method (when the parameter "#19710" = "0")

(1) Press the menu [String search].

(2) Set the character string to be searched for, and press the [INPUT] key.
(Example) G20 [INPUT]

→ The character string is searched downward from the character after the current cursor position.
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.
(Example) G20/MR :G20 are marked in red.
(Refer to "Marking function" described later for further details.)

(3) To continue searching, press the [INPUT] 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.

(2) During mass-editing, the marking cannot be made.

Operation method (when the parameter "#19710" = "1")

■For downward search

(1)	Press the menu [String search].	The sub menus of string search are displayed and [Down search] menu is highlighted. The cursor is displayed in the input area below.
(2)	Set the character string to be searched for, and press the [INPUT] key. (Example) G20 [INPUT]	The character string is searched downward from the character after the current cursor position. If the character string is found, 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. (Example) G20/MR :G20 are marked in red. (Refer to "Marking function" described later for further details.)
(3)	To continue searching, press the [INPUT] key again.	The next corresponding character string is searched for. When the end of the program is reached, the message "OK to search from the top?(Y/N)" appears. When the search is performed from the start again, press the [Y] or [INPUT] key. To end the search, press the [N] key. The string search is finished automatically and the screen returns to the main menu. The character string in the input area is deleted.
(4)	To change the search direction, press the menu [Upward search].	The search direction is changed upward.
(5)	To end the search, press the menu [Quit search] or ◁ key.	The string search is finished and the screen returns to the main menu. The character string in the input area is deleted.

■For upward search

(1) Press the menu [String search].
The sub menus of string search are displayed and [Down search] menu is highlighted.
The cursor is displayed in the input area below.

(2) Press the menu [Upward search].
The [Upward search] menu is highlighted.

(3) Set the character string to be searched for, and press the [INPUT] key.
(Example) G20 [INPUT]
The character string is searched upward from the character before the current cursor position.
If the target character string is found, 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.
(Example) G20/MR :G20 are marked in red.
(Refer to "Marking function" described later for further details.)

(4) To continue searching, press the [INPUT] key again.
The next corresponding character string is searched for.
When the start of the program is reached, the message "OK to search from the bottom?(Y/N)" appears.
When the search is performed from the end again, press the [Y] or [INPUT] key.
To end the search, press the [N] key. The string search is finished automatically and the screen returns to the main menu. The character string in the input area is deleted.

(5) To change the search direction, press the menu [Downward search].
The search direction is changed downward.

(6) To end the search, press the menu [Quit search] or ◁ key.
The string search is finished and the screen returns to the main menu. The character string in the input area is deleted.

Note

(1) "Executing" is displayed in flickering during string search.
(2) The setting of search direction is held until the power is turned OFF. The setting after the power is turned ON again is [Down search].
(3) During mass-editing, the marking cannot be made.

Marking function

Character strings to be searched for can be marked in colors.

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. All displayed searched character strings are marked in red.	Red
Search character string/MB	The cursor moves to the character string searched for. All displayed searched character strings are marked in blue.	Blue
Search character string/MG	The cursor moves to the character string searched for. All displayed searched character strings 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.

(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.

(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.

(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.

(5) With the multi-program display type, the character string and color to be marked are not synchronized in the each editing area.

(6) The marked status is maintained until the NC power is turned OFF.

(7) Marking function is invalid in mass-editing.

Important points when performing search

(1) If the designated character string cannot be found, the operation message "Designated character string not found" appears.

(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".)

3.6.8 Replacing Character Strings

A character string in a program can be searched for and replaced.

The following three 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 [INPUT] key.)
- All applicable character strings in the program are replaced in a batch. (Designate "/G" when setting.)
- The program character strings from the current cursor position to the last line are to be replaced in a batch. (Designate "/E" when setting.)

Operation method

(1) Press the menu [String replace].

(2) Designate the character string to be searched and replaced.
(Example) Searching for "G02" to replace with "G03"
G02/G03 [INPUT] (*1)
→ The character string is searched downward from the next character where the cursor is currently located. When the character string is found, the cursor moves to the first character of that string, and the operation message "Replace?(Y/N)" appears.
If any character string is not found in the program, the cursor does not move.

(3) Press the [Y] or [INPUT] key.
Press the [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 operation has been completed.)
The operation message "Designated character string not found" appears when the end of the program is reached.
"Editing" appears if replace is performed.

(4) Press the [Y] or [INPUT] key again if continuing the search/replace.
→ The same operation as above is performed.

(5) To end search/replace, press the [String replace].
→ The menu highlight returns to normal, and the character string in the input area disappears.
The replacement mode is held until the menu key is pressed.
(The character string remains in the setting area.)

(*1) When batch replacing for all the character strings in the program, add "/G" to the above setting.
Character string to be searched/character string to be replaced/G

(Example) 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

(1) 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.
Character string to be searched/character string to be replaced/E
(Example) G02/G03/E

(2) With the multi-program display type, the replacement on the side which is not being edited is not performed.

(3) "Executing" is displayed in flickering during the replacement.

(4) During mass-editing, the character string cannot be replaced.

3.6.9 Copying and Pasting Data

Operation method (Copying a line)

(1) Press the menu [Line copy].

(2) Designate the copy range.
(Example) From line 8 to line 10
8/10 [INPUT]
When copying 1 line, it is also possible to make a selection using the [↑] or [↓] key and designate by pressing the [INPUT] key.
➡ The background color for the range to be copied turns light blue.

Note

(1) If [INPUT] 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, or turned back to the previous saved status by pressing the menu [Undo] after the Line copy menu is pressed, the data copied in the line is canceled.
(4) During mass-editing, the string copy cannot be performed.
(5) 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
(6) 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. The data will be discarded after screen transition.

Operation method (Copying character)

(1) Move the cursor to the head character to be copied.

(2) Press the menu [Char copy].

(3) Move the cursor to the end character to be copied, and press the [INPUT] key.
➡ The background color for the range to be copied turns light blue. The range is fixed by pressing the [INPUT] key.

Note

(1) Characters of up to 500 bytes can be copied at a time. When the size of the selected characters exceeds 500 bytes, the message "Can not copy more" appears and the cursor does not move.
(2) The Copying will be canceled, if the [↑], [↓], [←], [→], [|←] or [→|] key is pressed before pressing the [INPUT] key.
(3) Light blue background for copy range will disappear when editing operation is resumed.
(4) As long as the original file is opened, the copied content will be saved even if you change the display type or shift the display left and right side. The data will be discarded after screen transition.

Operation method (Pasting the copied data)

(1) Move the cursor to the line where the data is to be pasted.

(2) Press the menu [Paste].

<When copying a line data>
The copied data is inserted upward on the line where the cursor is positioned.
<When copying characters>
The copied data is inserted in front of the cursor position.

<Note>
- In the overwrite mode, the data is inserted in front of the cursor position as well.

(3) Press the [INPUT] key.

Note

(1) With multi-program display type, data can be copied and pasted between the each editing areas by pressing the [<--> change] key.

(2) "Executing" is displayed in flickering during pasting.

(3) During mass-editing, copied data in the regular editing mode cannot be pasted.

(4) EOB is not converted to line feed code only when the character including EOB is copied with the program opened in mass-editing and pasted to the program in mass-editing. EOB is pasted converted to line feed code even if EOB has a comment in the following cases.

- When copied from the mass-editing screen and the pasted to regular editing screen
- When copied from the regular editing screen and pasted to the regular editing screen
- When copied from the regular editing screen and pasted to the mass-editing screen

3.6.10 Undoing Program Changes

This is used to return the program to the form when [INPUT] 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

(1) Change the operation to memory mode, and then press the main menu [Edit].

(2) Move the cursor to the position to be corrected, and input a character.
(Example) 135
If [Undo] is pressed during "Editing", the system will be returned to the state saved last.

(3) Press the [INPUT] key. → The edited machining program is saved to the device. The message "Editing" disappears.

(4) Press the menu [Undo]. → 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.

(5) Press the menu [Undo] again. → The contents of the file return to the same condition as when the file was saved in the procedure (3). These contents of the file are saved to the device.

Note

(1) This function is only valid when parameter "#8910 Edit undo" is set to "1".
(2) When returned to the previous status, the cursor will move to the head line.
(3) With the multi-program display type, returning to the previous contents is valid only in the active area.
(4) Undoing is not possible in mass-editing mode.
(5) When undoing is performed, the copied data will be canceled.
(6) "Executing" is displayed in flickering during undoing.
(7) When the parameter"#8939 Undo confirm msg" is set to "1", if [Undo] is pressed, the menu is highlighted and the operation message "OK?(Y/N)" appears. The program returns to the previous status by pressing [Y] or [INPUT] key. If other key is pressed, the operation is canceled. When the operation is canceled, the menu highlight returns to normal and the message is deleted.
(8) When the program does not return to the previous status by pressing [Undo], [Undo] is not highlighted and the operation message "Data protect" or "Executing automatic operation" appears.

3.6.11 Searching T codes

T codes (e.g T03, T0101) in a machining program can be searched.
You can search T codes both upward and downward.

(1) Press the menu [Down Tsearch].

The T code is searched downward from the next character where the cursor is currently located. When the T code is found, the cursor moves to the position of the T code.
The message "Designated T code not found" appears if there is no designated T code in the program.

(2) <To search downward consecutively>
Press the menu [Down Tsearch].

The next T code is searched.
When the search is reached the end of the program, the message "OK to search from the top?(Y/N)" appears.
When the search is performed from the start again, press the [Y] or [INPUT] key.
To end the search, press the [N] key.

<To search upward>
Press the menu [Upward Tsearch].

The character string is searched upward from the next character where the cursor is currently located. When the T code is found, the cursor moves to the position of the T code.
When there is no designated T code on the upper area where the cursor is currently located, the message "OK to search from the bottom?(Y/N)" appears.
When the search is performed from the bottom again, press the [Y] or [INPUT] key.
To end the search, press the [N] key.

Note

(1) The T code included in the comment is not searched.
The description which meets the following conditions is regarded as a T code and is to be searched.
- The description before "T" must not be an alphabetical character.
- The description which follows "T" must be any of "#", "[", "+", or "-".

(2) If the searching takes time, "Executing" is displayed in flicking every one second.

3.6.12 Searching a Program from Edit Screen

The menu [Program search] is displayed by setting the parameter "#1759 cfgPR09/bit2" (Operation search from edit screen) to "1", and you can use this function.
This function allows you to search for a program from the program edit window on the monitor screen or from the edit screen, and open it together with the current program. Hereafter, this function is referred to as "Operation search from edit screen".

There are two methods to perform "Operation search from edit screen".

- Press the [↑] or [↓] key to search for a program before or after the currently opened program and open it together with the current program.
- Press the [INPUT] key to search for the program with the name specified in the input section and open it together with the current program.

Performing "Operation search from edit screen" using the [↑] or [↓] key

When searching with the [↑] or [↓] key, a search is performed each time the key is pressed and searched program is opened together with the current program on the edit screen.
The program located before or after the currently displayed program in the same device and directory is searched for.

(Example) When the "004" program among the consecutive programs "001" to "006" stored in the NC memory is displayed in the active edit area on the edit screen

(1)	Press the menu [Program search].	➡	The cursor on the program disappears to move to the input section.
(2)	Press the [↑] or [↓] key to select a program to be searched for.	➡	Pressing the [↑] key searches for the previous program "003". Pressing the [↓] key searches for the next program "005". The message "Search completed" appears and the searched program is opened.
(3)	Press the menu [Program search].	➡	The cursor on the input section disappears to return to the program area.

Note

(1) The program that is located before and after the currently opened program in the order shown in the table below can be searched for using the [↑] or [↓] key.

The device where the displayed program is stored	Program order
Memory, memory 2	1. Ascending order by program number (numeric) 2. Ascending order by program number (character code)
HD	1. Ascending order by program number (character code)
Memory card, DS, USB memory	1. Ascending order by program creation date

(2) When "Operation search from edit screen" is performed, the sorting order set in the program list for operation search etc. is ignored.

(3) When "Operation search from edit screen" is performed, the following files in the same device and directory as the currently displayed program are skipped.

- Program protected by edit lock B, C
- Program protected by program display lock
- Folder

(4) If a large number of programs are stored on external device such as DS or USB memory, it may take a long time to search for a program in the device using the [↑] or [↓] key.

Performing "Operation search from edit screen" by designating the program name

The program stored in the same device and directly as the currently displayed program can be opened. Press the [INPUT] key to search for the program specified in the input section and open it together with the current program.

(Example) When the consecutive programs "001" to "006" are stored in the NC memory

(1)	Press the menu [Program search].	➡	The cursor on the program disappears to move to the input section.
(2)	Input the program name "006" to the input section, then press the [INPUT] key.	➡	The program "006" is searched for. The message "Search completed " appears and the searched program is opened.
(3)	Press the menu [Program search].	➡	The cursor on the input section disappears to return to the program area.

Note

(1) When a non-existent program name is designated, the message "Search error" appears. However, the value of the input section is not cleared.

(2) When any of the following files is designated, the message "Search error" appears. However, the value of the input section is not cleared.

- Program protected by edit lock B, C
- Program protected by program display lock

(3) When a folder name is designated, the message "Setting error" appears. However, the value of the input section is not cleared.

(4) "Operation search from edit screen" function does not allow you to perform searches that specify sequence number or block number. The message "Setting error" appears.

3.7 Program Editing Support Function

Edit – 

3.7.1 Displaying Input Mistakes

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. (Example) N01 G0 X100 Y50.; "X100" displayed as warning

The addresses to check no decimal point input are shown below.

(○:Check performed, -: No check performed)

Addresses	Decimal point command	Application	M/L	Decimal point check
A	Enabled	Coordinate position data		○ Only for axis name
	Disabled	Miscellaneous function code		
	Enabled	Geometric linear angle data		
	Disabled	Rotation table	M	
	Disabled	MRC program number	L	
	Disabled	Data settings, axis number	L	
	Enabled	Deep hole drilling cycle (2) Safety distance	L	
B	Enabled	Coordinate position data		○ Only for axis name
	Disabled	Miscellaneous function code		
	Disabled	Rotation table	M	
C	Enabled	Coordinate position data		○ Only for axis name
	Disabled	Miscellaneous function code		
	Enabled	Corner chamfering amount (,C)		
	Disabled	Rotation table	M	
	Enabled	Program tool compensation input	L	
	Enabled	Nose R compensation amount (Incremental)	L	
D	Disabled	Compensation number (tool position, tool radius compensation)	M	-
	Enabled	Auto tool length measurement, deceleration ranged	L	
	Enabled	Data settings	L	
	Disabled	Sub-program storage device number (,D)	M	
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		-
	Disabled	Compensation number (tool position, tool length compensation)	M	
I	Enabled	Circular center coordinates		○
	Enabled	Vector component for tool radius compensation/nose R compensation		
	Enabled	Custom fixed cycle hole pitch	M	
	Enabled	1st cut amount for deep hole drilling cycle (2)	L	

Addresses	Decimal point command	Application	M/L	Decimal point check
J	Enabled	Circular center coordinates		○
	Enabled	Vector component for tool radius compensation/nose R compensation		
	Enabled	Custom fixed cycle hole pitch, angle	M	
	Disabled	Dwell at deep hole drilling cycle (2) return point	L	
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	
	Disabled	Fixed cycle repeat count	M	
L	Disabled	Sub-program repeat count		-
	Disabled	Fixed cycle repeat count	M	
	Disabled	Program tool compensation input type selection (L2, L10, L11)	L	
	Disabled	Data setting selection (L70)	L	
M	Disabled	Miscellaneous function code		-
N	Disabled	Sequence No.		-
	Disabled	Data setting, data number	L	
O	Disabled	Program No.		-
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	

Addresses	Decimal point command	Application	M/L	Decimal point check
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	
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	
	Enabled	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		○ Only for axis name
	Disabled	2nd miscellaneous function code	L	
	Enabled	Program tool compensation input	L	
V	Enabled	Coordinate position data		○ Only for axis name
	Disabled	2nd miscellaneous function code	L	
	Enabled	Program tool compensation input	L	
W	Enabled	Coordinate position data		○ Only for axis name
	Disabled	2nd miscellaneous function code	L	
	Enabled	Program tool compensation input	L	
X	Enabled	Coordinate position data		○ Only for axis name
	Enabled	Dwell time		
	Disabled	2nd miscellaneous function code	L	
	Enabled	Program tool compensation input	L	
Y	Enabled	Coordinate position data		○ Only for axis name
	Disabled	2nd miscellaneous function code	L	
	Enabled	Program tool compensation input	L	

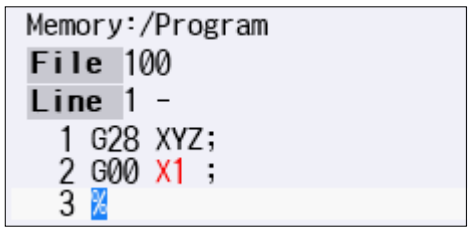
Addresses	Decimal point command	Application	M/L	Decimal point check
Z	Enabled	Coordinate position data		○ Only for axis name
	Disabled	2nd miscellaneous function code	L	
	Enabled	Program tool compensation input	L	

Performing a consecutive input mistake check when creating a program

(1) Press the menu [Open (New)] , input a new file name, and press the [INPUT] key. → Only an EOR program is created.

(2) Press the menu [Miss warning].

(3) Edit the program. → A warning is displayed at the moment the [INPUT] key is pressed.



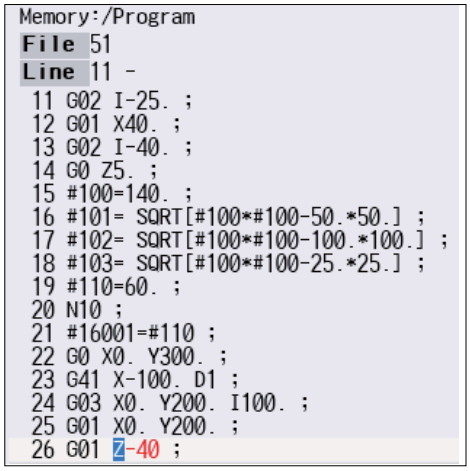
(4) Correct the input mistake and continue with program creation.

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 menu [Edit].

(3) Press the menu [Miss warning]. → 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.



(4) Press the menu [Next miss]. → The cursor moves to the next input mistake position.

```
22 G0 X0. Y300. ;
23 G41 X-100. D1 ;
24 G03 X0. Y200. I100. ;
25 G01 X0. Y200. ;
26 G01 Z-40 ;
27 X50.,R10. ;
28 Y#101 ;
29 G02 Y50 X#101 R#100 ;
```

(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] when "X10;" is input.
			A warning is displayed upon pressing [INPUT] when the "X10.Y" "." is inserted in the case where "X10Y" is input.

If the menu [Miss warning] 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 [Next miss] is grayed out and cannot be selected when the menu [Miss warning] 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.
(Example: 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.
(Example: 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 operation message "Input miss was detected" appears. If no input mistake is found, the operation message "Input miss was not detected" appears.

(9) With the multi-program display type, the input mistake check of the non-active area is not performed. When the editable side is switched with the menu [<--> change] during the input mistake check warning function ON, the cursor moves to next input mistake position. The messages "Input miss was detected" and "Input miss was not detected" are judged referring to the cursor position on the side being edited.

(10) Input mistake check warning function is disabled in mass-editing.

(11) "Executing" is displayed in flickering during checking input mistakes.

(12) Decimal point command is enabled or disabled for each address depending on the G code type, but the input mistake check is independent of the G code type. Thus the mistake check is carried out while the state of whether the decimal point command is enabled or disabled is kept the same for each address. Refer to the table in the previous section for whether the decimal point command is enabled or disabled for each address.

3.7.2 Performing Operation Search at the Cursor Position

By pressing the menu [Cursor OP srch] when the device is in the normal status (not being in status such as editing or character searching, etc.), operation search of the selected part system is carried out starting.

(1) Select the part system that the cursor operation search is performed.

(2) Open the program on the edit screen. → The program edit window appears.

(3) Move the cursor to the block to be carried out the cursor operation search.

(4) Press the menu [Cursor OP srch]. → Operation search is carried out starting from the block that the cursor is positioned.
During the search "Searching" will display blinking.
After search is completed, the display returns to the monitor screen.

The state of each part system after cursor operation search differs depending on the combination of the setting values of the parameters "#1285 ext21/bit0" (Multi-part system program management), "#1285 ext21/bit1" (Switching of program search type) and "#8144 OP srch NB all".

#1285 ext21/bit0 Setting value	#1285 ext21/bit1 Setting value	#8144 Setting value	Program displayed on the edit screen	State of each part system after cursor operation search	
				Currently displayed part system	Other part systems
0 (Invalid)	-	-	-	Cursor operation search is performed.	No operation is performed.
1 (Valid)	0 (for each part system)	-	-		
	1 (for all part systems at once)	0 (only for displayed part systems)	Same as the program where operation search is performed in the currently displayed part system		
			Different from the program where operation search is performed in the currently displayed part system		Operation search is performed at the head of the program displayed on the edit screen.
		1 (for all part systems at once)	Same as the program where operation search is performed in the currently displayed part system		The program searched in each part system is held and operation search is performed with the sequence No./block No. at the cursor position. (*1)
			Different from the program where operation search is performed in the currently displayed part system		Operation search is performed with the program No. displayed on the edit screen and with the sequence No./ block No. at the cursor position.

(*1) When the searched program is in a different part system from the program displayed on the edit screen, the message "Search error" appears.

Note

(1) During program editing, the menu [Cursor OP srch] is disabled and the cursor operation search cannot be performed.

(2) During the MDI program is displayed, the menu [Cursor OP srch] is disabled and the cursor operation search cannot be performed.

(3) During automatic operation, checking or restart search by selected part system, the cursor operation search cannot be performed displaying the messages "Executing automatic operation", "Program checking" and "Program restarting" for each status.
In such cases, the display does not transit to the monitor screen. However, it is enabled when the parameter "#11039 Cusr pos srch typ" is set to "1" even if the operation is under single block stop. For details, refer to "Search during single block stop" in "10.9.1 Program Editing".

(4) During automatic operation, checking or restart search by other than the selected part system, the cursor operation search can be performed, and the messages "Executing automatic operation", "Program checking" and "Program re-starting" will not appear. In this case, the display transits to the monitor screen.

(5) The cursor position search is not available for the sub part system I or II in M80V/M80VW Series, or the sub part system II in M800V Series. This menu is non-selectable when the search cannot be operated.

(6) When the parameter "#8144 OP srch NB all" is "1", if the search error occurs in the displayed part system, the message appears and the search is not performed in any part system.

3.7.3 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 N99999999.

This function is disabled during mass-editing.

Operation method

(1) Press the menu [N auto add].

(2) Input the incremental value.
(Example) 10 [INPUT]

<When creating a new program>

(3) Press the menu [Open (New)], then press the menu [Memory].

(4) Input a file name not used, then press the [INPUT] key.
(Example) 101

(5) Input a program with N No.
(Example) N100G28XYZ

<When editing an existing program>

(3) Press the menu [Open], then press the menu [Memory].

(4) Select a file, then press the [INPUT] key.

(5) Move the cursor to the position where data is to be inserted. Press the [INSERT] key.

(6) Input " ; " (EOB).

(7) Input the second block data and " ; " (EOB).
(Example) G92X0.Y0.Z0.;

(8) Press 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.
(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.
(3) When "0" is input, this function is canceled.
(4) When an illegal value or 9 characters or more is input and [INPUT] is pressed, the "Setting error" appears.
(5) The incremental value can be set even when the machining program has been not opened. Even if the machining program is opened again, the incremental value is not invalid.
(6) The incremental value is not cleared with the screen transition.
(7) This function is enabled only during regular editing of edit screen.
(8) 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.
(9) Even if the same N No. exists by automatic addition, an error will not occur.
(10) The system separately has N No. in a right area and left area of the edit screen.
(11) 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 G:N110 28 X0 Y0 Z0 ;

N100 G28 X0 Y0 Z0 ;N200G90X20.Y20 .Z20.; -> Input ";" -> N100 G28 X0 Y0 Z0 :N110N200G90X20 .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 any comment has been added

(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 100000000.

(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 ;

3.7.4 G Code Guidance

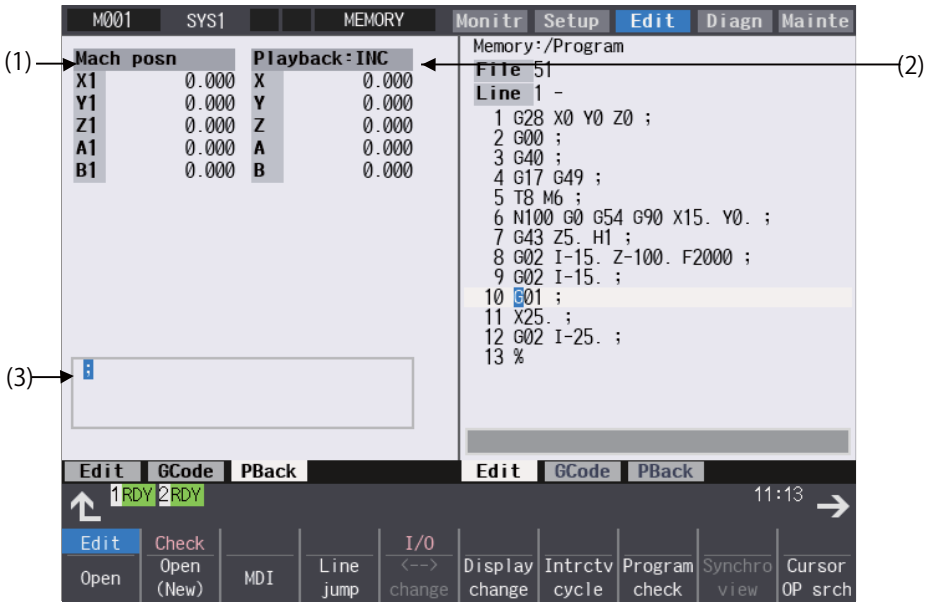
For details, refer to "2.5.3 G Code Guidance".

3.7.5 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.
The playback counter and valid address key (X, Y, U, V, etc.) are different depending on the parameter "#1076 AbsInc" (ABS/INC address) setting. A parameter "#1076 Abslnc" can be set individually for each part system.

	#1076 AbsInc (For L system only)	#1126 Playback G90	Reference
M system	(This setting is meaningless.)	0 (Incremental position)	Absolute command/incremental command depending on G command
		1 (Absolute position)	
L system	0 (Absolute command/incremental command with G command)	(This setting is meaningless.)	
	1 (Absolute command/incremental command with address code)		Absolute command/incremental command depending on address code

<Absolute command/incremental command with G command (M/L System)>



Display items

Display items		Details
(1)	Machine position counter	This displays the machine position counter. Up to max. number of axes in a part system can be displayed. M system: Max. eight axes L system: Max. five axes, the sixth axis or later is switched by the menu [Next axis]. When the rotary axis (except for all coordinate linear type), the display range is 0.000° to 359.999°.
(2)	Playback counter	This displays the playback movement amount (axis movement value). When the absolute command or incremental command depends on G command, the title and playback movement amount differs according to the parameter "#1126 Playback G90". (*1) When the absolute command or incremental command depends on address code, both titles are displayed. Up to max. number of axes in a part system can be displayed. M system: Max. eight axes L system: Max. five axes, the sixth axis or later is switched by the menu [Pback next ax]. When the rotary axis (except for all coordinate linear type), the display range is as below: For INC (Rotary type short-cut disabled, workpiece coordinate linear type): -359.999° to 359.999° For INC (Rotary type short-cut enabled: -180.000° to 179.999° For ABS: 0.000°to 359.999°
(3)	Playback editing input area	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.

(*1) The displayed title and playback movement amount are as follows.

#1126 Playback G90	Title	Playback movement amount
0 (Incremental position)	Playback: INC	Playback movement amount before axis moves + axis movement amount
1 (Absolute position)	Playback: ABS	The program position + manual interrupt amount

Axis name displayed on the playback counter:

#1266 ext02/bit0 Axis name extension valid	#1601 axnameEx Axis name extension letter	Axis name to be displayed
0 (Invalid)	-	This displays the axis name set by the parameter "#1013 axname".
1 (Valid)	Already set (Name extension axis)	This displays the combination of the axis name set by the parameter "#1013 axname" and the axis name extension character set in "#1601 axnameEx".
	Not set (Non-name extension axis)	This displays the axis name set by the parameter "#1013 axname".

Operation method (Absolute command/incremental command with G command)

(1) Set "#1126 Playback G90" to "0" on the parameter screen.

→ Incremental value mode is set.

(2) Press the main menu [Edit] on the edit screen, and press [Displaychange] or  key several times.

→ The "playback display type" is displayed.
The menu [<--> change] cannot be used.
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.

<Note>
- When the program is displayed on the right side, the playback edit is displayed on the right side.
- When the incremental value is set, playback movement amount will be cleared to "0" for all axes.

(4) Move the axis during manual mode.

→ The axis movement amount is displayed in playback movement amount.
When the absolute position is set, "the program position + manual interrupt amount" is displayed as the playback movement amount.
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.

<Note>
- If the [C.B] key is pressed, all the data in the playback editing area is deleted.
- When the [↑] or [↓] key is pressed, the program cursor in the program editing area will move.
- When the page up key or page down key is pressed, the program in the program editing area will scroll.
- When the menu key is pressed, the process corresponding to the program editing area will run.
  When the cursor appears on input area of the program editing area by pressing [Line jump], [Line copy], [Line clear], [String search], [String replace], [MDI resist] or [N auto add], the key input is made to the input area of the program editing area.
  When window is displayed by pressing [Open], [Open (New)] or [Erase file], the key input is made to the currently displayed window.
- The program cursor in the program editing area cannot move left and right, but can move only up and down.

(6) Press the axis address key such as [X] or [Y].

→ 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.

<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, 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 position, 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.
The program in the program editing area is saved.

Operation method (Absolute command/incremental command with address code)

(1) Confirm that the parameter "#1076 AbsInc" is "1".

(2) Edit a program the same as the procedure (2) to (7) of "Absolute command/incremental command with 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 program created by playback editing is not suffixed by "EOB(;)", it is automatically added by [INPUT].

(4) 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. The playback absolute counter has the G92 coordinate system preset released, and returns to "program position + manual interrupt amount". The playback increase counter is cleared to "0".

(5) 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.

(6) If the program being playback edited automatically starts, playback editing cannot be continued.

(7) The playback movement amount after playback editing will differ according to the parameter "#1126 Playback 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 position	For an incremental position
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.

(8) The playback editing is disabled during mass-editing.

(9) 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.)

(10) 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.

3.7.6 Optimal program conversion

This function converts two or more rotary axis commands to one rotary axis command, in a block of the machining program where the tool center point control is commanded. When the menu [Program Convert] is pressed, the program conversion screen is displayed.

The following effect can be obtained by the program conversion.

- When the reciprocation range of the rotary axis is within twice of the tolerable error amount, the reciprocation is eliminated and the cycle time is reduced.
- For the rotary axis with fine motion, the fine motion is eliminated and the surface quality is improved.

This function outputs machining programs optimized based on "Tolerable amount", "Precnv prog type", etc. entered on the program conversion screen.

For the file name of the program after conversion, "_CONV*" is added to the end of the program name before conversion.

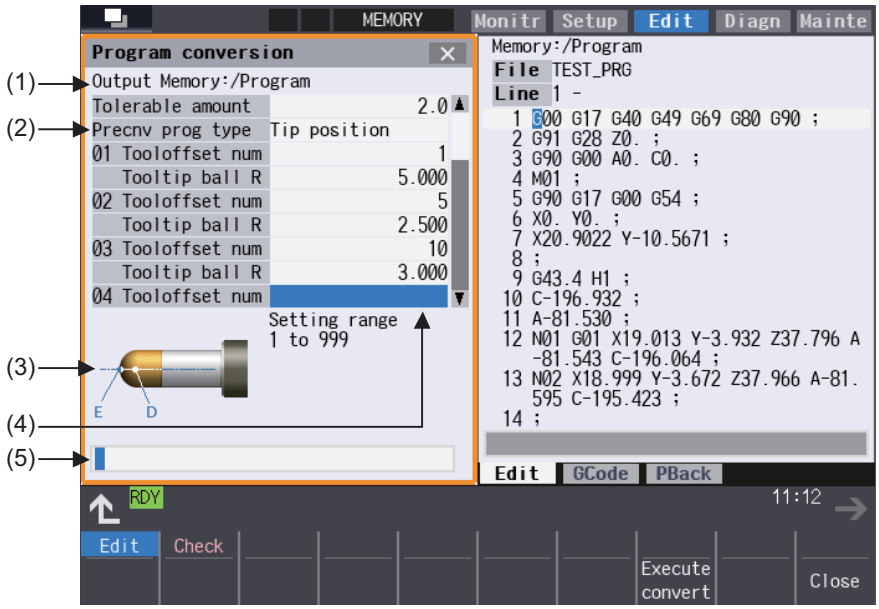
"*" indicates a value (01 to 50) which is ten times as large as the tolerable error amount.

Note

(1) This function cannot be used when the parameter "#1285 ext21/bit0" (Multi-part system program management) is "1" (Valid).

(2) The program name before conversion should be 25 characters or less.

(3) When the output destination is "Memory" or "Memory2" and the program name before conversion includes lower case alphabet, the program name after conversion is entirely upper case.



Display items

Display items	Details
(1) Output destination path	The output destination path of the converted program is displayed. The path set in "#19725 Prgcnv-output:dev" and "#19726 Prgcnv-output:dir" is displayed. When the path is too long, the portion exceeding the display area is not displayed. (Example) Memory:/program HD:/PRG/SUB

<table>
<tr><th colspan="2">Display items</th><th>Details</th></tr>
<tr><td>(2)</td><td>Program conversion setting items</td><td>The program conversion conditions are displayed/set.
<table>
<tr><th>Setting item</th><th>Details</th><th>Setting range</th></tr>
<tr><td>Tolerable amount</td><td>Set the tolerable error amount of angle when the program is converted.</td><td>0.1 to 5.0 (°)</td></tr>
<tr><td>Precnv prog type</td><td>Select the base position of the applied tool length compensation when the conversion program is created by CAM, etc.
When [Tip position] is selected, set "Tooloffset num" and "Tooltip ball R" to be used in the program. Up to 16 sets of tool information can be set.</td><td>1: Tool center
2: Tip position</td></tr>
<tr><td>Tooloffset num (01 to 16)</td><td>Set the tool compensation No. to be used in the program.</td><td>1 to 999</td></tr>
<tr><td>Tooltip ball R (01 to 16)</td><td>Set the tip ball radius of the tool set in "Tooloffset num".</td><td>0.001 to 9999.999 (*1)</td></tr>
</table>
(*1) #1003 iunit = B and #1041 I_inch = 0</td></tr>
<tr><td>(3)</td><td>Guide drawing</td><td>The input contents are displayed in the guide drawing.</td></tr>
<tr><td>(4)</td><td>Input guidance</td><td>The input guidance at the cursor position is displayed.</td></tr>
<tr><td>(5)</td><td>Input area</td><td>The input data is displayed.</td></tr>
</table>

Menus

Menus	Details
Tool center	This selects the base position of the tool length compensation applied to the program before conversion. These menus are displayed when the cursor is at "Precnv prog type". When [Tip posn] is selected, "Tooloffset num" and "Tooltip ball R" of the tool to be used in the program need to be set.
Tip posn	
Execute convert	This executes program conversion.
Close	This closes the pop-up window and quits this function.

Setting the conversion conditions

The converted program is output to the location specified with the parameters "#19725 Prgcnv-output:dev" and "#19726 Prgcnv-output:dir".

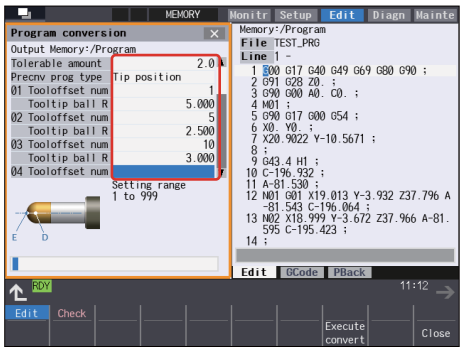
(1) Press the menu [Open] on the edit screen to open the machining program to be converted.

(2) Press the menu [Program Convert]. → The program conversion screen is displayed.

<Note>

♦If the menu [Program Convert] is pressed while a program that cannot be converted is open, "Selected file cannot be converted" is displayed and the program conversion screen is not displayed.

(3) Set the items displayed on the screen. → The set contents are displayed.



(4) Press the menu [Execute convert].

Precautions

(1) Do not edit the program before conversion during program conversion.

(2) The following commands are not the target of the program conversion. The blocks where these commands are used are copied without conversion.

♦Common variables
♦Local variables
♦System variables
♦Machine tool builder macro variables
♦Extended common variables
♦Operation commands
♦Control commands
♦External output commands
♦Two consecutive alphabet commands

(3) Do not perform program conversion again for the machining program after program conversion.

(4) Program conversion is performed for only the program that is open on the edit screen.
When program conversion is performed for a machining program to call subprogram, user macro, etc., only the source program is converted.

(5) When the CNC is turned OFF during program conversion, the machining program in the middle of the conversion may not be deleted and may remain in the output destination. Do not operate the machining program in the middle of the conversion.
The name of the machining program in the middle of the conversion differs depending on the output destination device.

♦When the output destination device is "HD", "_TMP" is added to the end of the program name before the conversion.
♦When the output destination device is not "HD", it begins with "~" and a 11-character number follows.

(6) During program conversion, cycle start is not available in any part systems. When the "Automatic operation "start" command" signal (ST) is input, the stop code (T01 0191) "Cycle start prohibit" occurs.

(7) When any of all the part systems is during automatic operation, restart search, or program check, program conversion cannot be performed.

(8) The format of one block in the conversion program should be "Prefix character string Axis address command Suffix character string ;". The prefix character string and the suffix character string must be within 63 characters respectively. If the number of characters exceeds the limitation or the command order of the axis address and the other items is different, conversion may not be performed correctly. The prefix character string and the suffix character string can be omitted.

(Example)

N101 G01	X-3.6 Y15.8 Z.9 A-49.4 C-11.7	F2000
Prefix character string	Axis address command	Suffix character string

(9) When the output destination device is "HD", set the parameters "#1926 Global IP address" and "#11005 PC IP address" to belong to the same subnet. If they are set to different subnets, the converted machining program is not output to the "HD" after optimal program conversion.

(10) When any of the following characters is included in the file name, operation search cannot be performed. Do not use the following characters.

\ / : , * ? " < > | space

(11) When the specification of HD mode operation is disabled for M800VW Series, do not set the parameter "#19725 Prgcnv-output:dev" to "3" (HD). If optimal program conversion is executed while it is set to "3", the program after conversion is output to the storage location of the program before conversion.

(12) When the parameter "#1760 cfgPR10/bit2" (Enable HD mode on IPC) is "0" for M80V Series, do not set the parameter "#19725 Prgcnv-output:dev" to "3" (HD). If optimal program conversion is executed while it is set to "3", the operation message "IPC connection error" is displayed and conversion is not performed.

(13) Only when "Precnv prog type" is set to "Tip position", 100 variables are stored in the order in the machining program. Command the variables in the format of "#Variable No. = Value".
The variables for only address H in the G43.4 command are considered. The block where variables other than H command is copied without conversion. (When the parameter "#1274 ext10/bit3" (Tool offset number H stand-alone command) is "1", the variable is considered also when the H stand-alone command is issued.)
Also when the tool compensation No. is commanded with the variable, the variable is stored to perform compensation of tool tip ball deviation.

(Example)
Machining program before conversion

```
#100 = 1 ;                                  ..... Command in the "#Variable No. = Value" format
#101 = 1000 ;                               ..... Command in the "#Variable No. = Value" format
      :
G43.4 H#100 ;                               ..... Compensated as G43.4 H1
G01 ;
A-81.000 C-196.000 F#101 (Not converted)
A-81.530
C-196.932
N01 X19.013 Y-3.932 Z37.796 A-81.543 C-196.064
N02 X18.999 Y-3.672 Z37.966          C-195.423
N03 X18.982 Y-3.412 Z38.137 A-81.516 C-194.776
      :
```

Machining program after conversion

```
#100 = 1 ;                                  ..... Command in the "#Variable No. = Value" format
#101 = 1000 ;                               ..... Command in the "#Variable No. = Value" format
      :
G43.4 H#100 ;                               ..... Compensated as G43.4 H1
G01 ;
A-81.000 C-196.000 F#101 (Not converted)    ..... The variable is used in other than G43.4, and
A-81.530                                          copied without conversion.
C-195.932
N01 X19.010 Y-3.931 Z37.796 C-195.932       ..... Tool center point compensation is performed
N02 X18.999 Y-3.672 Z37.964 C-195.423             based on tip ball radius of tool compensation
N03 X18.982 Y-3.412 Z38.137 C-194.776             number 1.
      :
```

The variables cannot be correctly stored in the following cases.

<When the variable is stored as "0">

- 1st commanded variable when 101th variable is commanded
- Variable referred to without assigning a value in the machining program
- Variable whose No. is 9 digits or more in address H of the G43.4 command

<When the variable is not stored>

- The format is not "#Variable No. = Value".
  (Example)
  #100 = 1; (It is updated to "#100 = 1" due to "#Variable No. = Value".)
  #100 = 1+1; ("#100" is not updated due to "#Variable No. = Operation commands".)
  G43.4 H#100; (It is judged as "#100 = 1".)
- Variable whose No. is designated by variable or whose name is defined by the SETVNn command
  (Example)
  #[#100] = 1; (The variable No. is designated by the variable.)
  [#HOSEI] = 2; (The variable name is used.)

The control commands (IF to GOTO, WHILE DO) are ignored. For the machining program where the variable is updated by the control commands, it is updated by the last sets of variables.

(Example)

```
 #100=5;
IF[conditional expression] THEN;
  #100=1;
ELSE;
  #100=2;
ENDIF;
      :(omitted)
G43.4 H#100; (It is judged as "#100 = 2".)
```

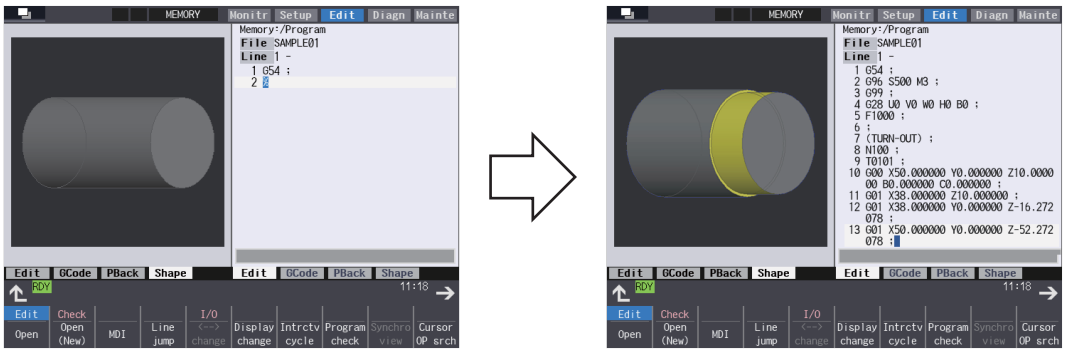
3.7.7 Displaying the Finished Shape

The finish shape view function displays the shape corresponding to the command of machining program input on the edit screen, and this function is intended to eliminate programming errors. You can easily confirm the machining shape without automatic operation or graphic check.
This function is an optional specification. To use this function, the setting by the MTB is required.

Press the menu [Displaychage] on the edit screen to switch to the "Shape" display type.

Confirming the machining shape when programming

(1) Press the menu [Open (New)] , input a new file name, and press the [INPUT] key.
(2) Each time you input a new block in the machining program, the finish shape is displayed in graphic drawing.



Confirming the machining shape with the existing program

(1) Press the menu [Open], select the machining program and then press [INPUT] key.
(2) When you press the menu [View Update], the input machining program is reflected in the machining shape.
(3) When you add a program block, it is reflected in the machining shape.

Note

(1) The finish shape view function cannot draw a serial or MDI program.
(2) Even when enlarging/reducing, moving, rotating, cross-section surface or automatic scale is operated while the workpiece is being displayed, the workpiece shape is not initialized.
(3) For the models other than M800VW Series, if you open the finish shape view, switch it to 3D program check, and then return to the finish shape view, the workpiece shape is initialized.
(4) When you select [Displaysetting] - [3 edit] or [4 edit] on 15-type or 19-type display, the machining program of the selected editing area is reflected to the finish shape.

3.8 Editing a Multi-part System Program

The parameter "#1285 ext21/bit0" (Multi-part system program management) sets this function valid or invalid.

This function can be used when NC memory or NC memory 2 is selected as a device.

3.8.1 Multi-part System Simultaneous Program Editing

When the parameter "#8977 Multi-$ simul edit" is set to "1" (valid), if you perform the operation (*1) to open a machining program in NC memory or NC memory 2 on the edit screen, the specified machining program of the displayed part system in the currently selected editing area and the machining program (*2) of the same name in another part system in the deselected editing area are opened simultaneously in the right and left panes.

(*1) To open a machining program, follow the steps shown below. The details on each operation are described in the table later.

- Select menu [Edit] - [Open] to open an existing machining program.
- Select menu [Edit] - [Open (New)] to create and open a new machining program.
- The parameter "#8938 Edit-Not show Prg" is set to "0" and the display changes to the edit screen after operation search or check search has been carried out. Then, the searched machining program is opened automatically.
- Execute operation search from edit screen.
- Execute operation search from edit screen on the program edit window on the monitor screen when the parameter "#8938 Edit-Not show Prg" is set to "0".

(*2) The machining program that satisfies the following conditions is opened in the deselected editing area.

- Machining program in another part system of the same name as for the machining program that is to be opened in the currently selected editing area
- If any machining program that satisfies machining program conditions is not registered in the smallest part system other than the currently selected part system, no machining program is opened in the deselected editing area. Then, the machining program opened in the deselected editing area is closed.

Note

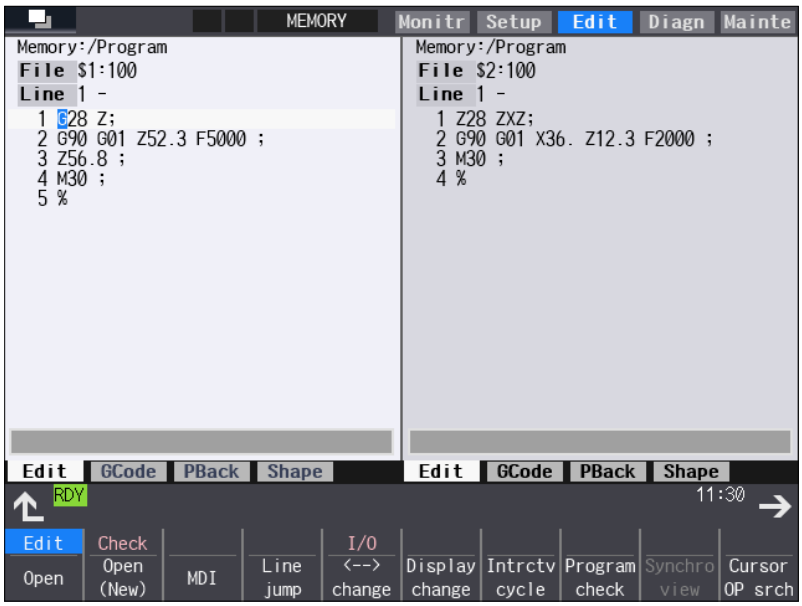
(1) If a machining program in NC memory or NC memory 2 is opened while the parameter "#8977 Multi-$ simul edit" is set to "1" (valid), the machining program opened in the deselected editing area is closed.

(2) The multi-part system simultaneous program editing is also available for MTB macro program and MDI.

(3) When the machining program for multi-part system simultaneous program editing is opened while the display type is not set to the multi-program display type, if the display type is changed to the multi-program display type, the machining program that is in another part system, of the same name as for the machining program opened in the currently selected editing area is displayed in the deselected editing area.

Operation to select the menu [Edit] - [Open] or to execute operation search from edit screen and open an existing machining program (Case in which the 1st part system is set as the displayed part system)

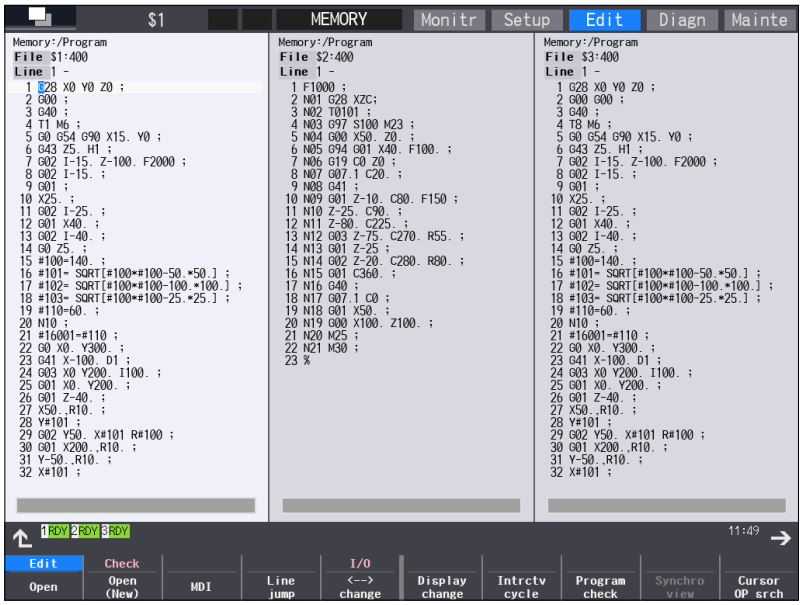
<For 2-program simultaneous editing>



○: Machining program registered ×: No machining program registered

$1	$2 (*1)	Operation
○	○	A programs in the 1st part system is displayed in the currently selected editing area, and that in the 2nd part system is simultaneously displayed in the deselected editing area.
○	×	This displays a machining program in the 1st part system in the currently selected editing area. Then, if the machining program is opened in the deselected editing area, it is closed.
×	○	A confirmation message (*2) is then displayed. Selecting "Y" creates and displays a new empty file in the 1st part system in the currently selected editing area. Machining programs in the 2nd part system are displayed in the deselected editing area.
×	×	If the designated machining program does not exist in all part systems, the message "Designated file does not exist" appears, that machining program is not displayed.

<For 3-program simultaneous editing> (15-type/19-type)



○: Machining program registered ×: No machining program registered

$1	$2 (*1)	$3 (*1)	Operation
○	○	○	A machining program in the 1st part system is displayed in the currently selected editing area, and those in the 2nd and 3rd part systems are simultaneously displayed in the deselected editing areas. The position to open the machining program in the 2nd and 3rd part system will be decided depending on the position of the currently selected editing area; on the left, in the middle or on the right. (The machining program will be opened in empty area from the left in ascending order of the part systems. Even though the G code guidance is displayed, the machining program will be opened behind the G code guidance.)
○	×	×	This displays a machining program in the 1st part system in the currently selected editing area. Then, if the machining program is opened in the deselected editing area, it is closed.
○	○	×	This displays a machining program in the 1st part system in the currently selected editing area. This displays a machining program in the 2nd or the 3rd part system in deselected editing area. Then, if the machining program is opened in another deselected editing area, it is closed.
○	×	○	
×	○	×	A confirmation message (*2) is then displayed. Selecting "Y" creates and displays a new empty file in the 1st part system on the currently selected editing area. A machining program in the 2nd or the 3rd part system is displayed in the deselected editing area. Then, if the machining program is opened in another deselected editing area, it is closed.
×	×	○	
×	○	○	A confirmation message (*2) is then displayed. Selecting "Y" creates and displays a new empty file in the 1st part system on the currently selected editing area. Machining programs in the 2nd and the 3rd part systems are displayed in the deselected editing areas. Then, if the machining program is opened in another deselected editing area, it is closed.
×	×	×	If the designated machining program does not exist in all part systems, the message "Designated file does not exist" appears and that machining program is not displayed.

(*1) For part systems other than the 1st part system, the 2nd or the 3rd part system is described as a typical one.

(*2) The operation message "Create new file?(Y/N)" appears. If "N" is selected, the message "A file does not exist" appears, and the target file is not opened. If "Y" is selected while the remaining number of registered machining programs is "0", the message "Program entry over" appears, and no machining program is displayed.
When the parameter "#1285 ext21/bit2" (Multi-part system program generation and operation) is ON, "Create new file?(Y/N)" does not appear and the file will not be opened.

Operation to select the menu [Edit] - [Open (New)] and open an existing machining program (Case in which the 1st part system is set as the displayed part system)

Multi-part system program generation and operation (#1285 ext21 bit2)	Operation
ON (Newly creates only a machining program in the displayed part system.)	This displays machining programs in the 1st part system in the currently selected editing area. When machining programs exist in the 2nd part system, they are simultaneously displayed in the deselected editing area. (*1) For 3-program simultaneous editing, machining programs in the 2nd and the 3rd part systems are simultaneously displayed in the deselected editing areas. (*1) When no machining programs exist in the 2nd part system, the machining programs opened in the deselected editing area are closed. (*1)
OFF (Collectively creates new machining programs in all the part systems.)	When no machining programs exist in all the part systems, a machining program in the 1st part system is displayed in the currently selected editing area and a machining program in the 2nd part system is displayed in the deselected editing area simultaneously. (*1) For 3-program simultaneous editing, machining programs in the 2nd and the 3rd part systems are simultaneously displayed in the deselected editing areas. (*1) When the machining programs in the 2nd part system exist, the message "Designated file already exists" appears and the file cannot be opened. (*1)

(*1) For part systems other than the 1st part system, the 2nd or the 3rd part system is described as a typical one.

Operation to automatically open the searched machining program (When the 1st part system is set as the displayed part system)

<For 2-program simultaneous editing>

Currently selected editing area	Deselected editing area	Operation
A machining program is not opened.	A machining program is not opened.	A machining programs in the 1st part system is displayed in the currently selected editing area, and that in the 2nd part system is simultaneously displayed in the deselected editing area. (*1)

For the conditions other than the above, no operation is performed.

<For 3-program simultaneous editing> (15-type/19-type)

Currently selected editing area	Deselected editing area 1	Deselected editing area 2	Operation
A machining program is not opened.	A machining program is not opened.	A machining program is not opened.	The currently selected editing area shows a machining program of the 1st part system. The deselected editing areas show machining programs in ascending order of the part systems from the left.

For the conditions other than the above, no operation is performed.

(*1) For part systems other than the 1st part system, the 2nd or the 3rd part system is described as a typical one.

Note

(1) If the searched machining program is not registered in the displayed part system, it is not opened. When the searched machining program is registered only in the displayed part system, no file is opened in the deselected editing area.

Operation method (Opening the existing machining programs of the same name in all part systems (When the 1st part system is set as the displayed part system))

Perform the following procedure to open the machining program, "100", under the following conditions.

- The parameter "#8977 Multi-$ simul edit" is set to "1" (valid).
- The machining program, "100", is registered in all part systems of NC memory.

(1)	Press the menu [Open].	→	The list appears as a pop-up window.
(2)	Use the [↑], [↓], [←], and [→] keys to align the cursor with the target machining program (100), and press the [INPUT] key. The same operation can also be performed by inputting the machining program name (100) to be edited in the input section and pressing the [INPUT] key.	→	The machining program "100" in the 1st part system is displayed in the currently selected editing area, and the machining program "100" in the 2nd part system is displayed in the deselected editing area.

Operation method (Opening the existing machining programs of the same name in the 1st and 4th part systems (When the 1st part system is set as the displayed part system))

Perform the following procedure to open the machining program, "200", under the following conditions.

- The parameter "#8977 Multi-$ simul edit" is set to "1" (valid).
- The machining program, "200", is registered in the 1st and 4th part systems of NC memory.

(1)	Press the menu [Open].	→	The list appears as a pop-up window.
(2)	Use the [↑], [↓], [←], and [→] keys to align the cursor with the target machining program (200), and press the [INPUT] key. The same operation can also be performed by inputting the machining program name (200) to be edited in the input section and pressing the [INPUT] key.	→	The machining program "200" in the 1st part system is displayed in the currently selected editing area, and the machining program "200" in the 4th part system is displayed in the deselected editing area.

Operation method (Opening the existing machining programs only in the 1st part system (When the 1st part system is set as the displayed part system))

Perform the following procedure to open the machining program, "300", under the following conditions.

- The parameter "#8977 Multi-$ simul edit" is set to "1" (valid).
- The machining program, "300", is registered in the 1st part system of NC memory.

(1)	Press the menu [Open].	→	The list appears as a pop-up window.
(2)	Use the [↑], [↓], [←], and [→] keys to align the cursor with the target machining program (300), and press the [INPUT] key. The same operation can also be performed by inputting the machining program name (300) to be edited in the input section and pressing the [INPUT] key.	→	The machining program "300" in the 1st part system is displayed in the currently selected editing area, and the machining program opened in the deselected editing area is closed.

Operation method (Opening the existing machining programs only in the 3rd part system (When the parameter "#1285 ext21/bit2" is set OFF and the 1st part system is set as the displayed part system))

Perform the following procedure to open the machining program, "400", under the following conditions.

- The parameter "#8977 Multi-$ simul edit" is set to "1" (valid).
- The machining program, "400", is registered in the 3rd part system of NC memory.

(1) Press the menu [Open]. → The list appears as a pop-up window.

(2) Use the [↑], [↓], [←], and [→] keys to align the cursor with the target machining program (400), and press the [INPUT] key.
The same operation can also be performed by inputting the machining program name (400) to be edited in the input section and pressing the [INPUT] key. → The operation message "Create new file?(Y/N)" appears.

(3) Press the [Y] key. → The machining program, "400", is newly created in the 1st part system.
The machining program "400" in the 1st part system is displayed in the currently selected editing area, and the machining program "400" in the 3rd part system is displayed in the deselected editing area.

Operation method (Creating a new machining program (When the parameter "#1285 ext21/bit2" is set ON and the 1st part system is set as the displayed part system))

Perform the following procedure to create a new machining program, "500", under the following conditions.

- The parameter "#8977 Multi-$ simul edit" is set to "1" (valid).
- The parameter "#1285 ext21/bit2" is set to "1" (to newly create only machining programs in the displayed part system).

(1) Press the menu [Open (New)]. → The list appears as a pop-up window.

(2) Input the machining program name (500) to be newly created in the input section, then press the [INPUT] key. → The machining program "500" in the 1st part system is displayed in the currently selected editing area, and the machining program opened in the deselected editing area is closed.

Operation method (Creating a new machining program (When the parameter "#1285 ext21/bit2" is set OFF and the 1st part system is set as the displayed part system))

Perform the following procedure to create a new machining program, "600", under the following conditions.

- The parameter "#8977 Multi-$ simul edit" is set to "1" (valid).
- The parameter "#1285 ext21/bit2" is set to "0" (to collectively create new machining programs in all part systems).

(1) Press the menu [Open (New)]. → The list appears as a pop-up window.

(2) Input the machining program name (600) to be newly created in the input section, then press the [INPUT] key. → The machining program "600" in the 1st part system is displayed in the currently selected editing area, and the machining program "600" in the 2nd part system is displayed in the deselected editing area.

Operation to be performed when the menu [Open], [Open (New)], or [MDI] is pressed while editing a machining program

If the menu [Open], [Open (New)], or [MDI] is pressed during editing of a machining program while the parameter "#8977 Multi-$ simul edit" is set to "1" (valid), whether to save both the contents of the currently selected editing area and those of the deselected editing area is confirmed regardless of whether the machining program to be opened actually is targeted for multi-part system simultaneous program editing.

Currently selected editing area	Deselected editing area	Operation
Not editing	Not editing	The operation of the pressed menu is performed.
Editing	Not editing	(1) The operation message "Save current file?(Y/N)" is displayed. (2) After any key has been pressed, the operation of the pressed menu is performed.
Not editing	Editing	(1) The message "Save right side file?(Y/N)" ("Save left side file?(Y/N)", "Save center file?(Y/N)") is displayed. (2) After any key has been pressed, the operation of the pressed menu is performed.
Editing	Editing	(1) The operation message "Save current file?(Y/N)" is displayed. (2) After any key has been pressed, the operation message "Save right side file?(Y/N)" ("Save left side file?(Y/N)", "Save center file?(Y/N)") is displayed. (3) After any key has been pressed, the operation of the pressed menu is performed.

3.8.2 Special Program Editing Display for Synchronization between Part Systems

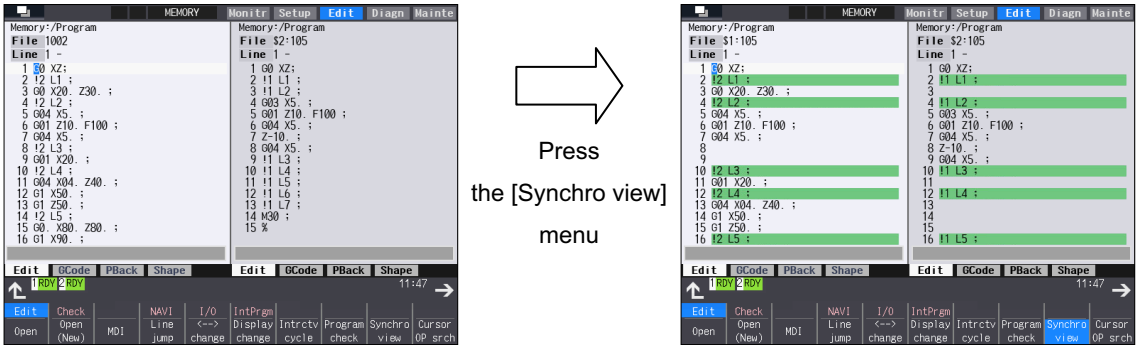
Edit – 

Pressing the menu [Synchro view] enables synchronized display of the left- and right-side programs aligned when the following conditions are satisfied:

- The parameter "#1285 ext21/bit0" (Multi-part system program management) is set to "1" (valid).
- Multi-program display type is selected.
- The left and right edit areas are displaying the same named programs of different part systems stored on NC memory or NC memory 2.

The following timing synchronization symbols can be used simultaneously.

- "!"
- Timing synchronization M codes specified by the parameters (#1310 and #1311)
- M/S/T/G codes specified by the timing synchronization symbol definition files for synchronous view (created by the MTB)



Note

(1) The synchronous display is also available for MTB macro program and MDI stored on NC memory.

(2) Editing can be performed in the selected editing area, and the synchronous display is disabled when "Editing" is being displayed. The synchronous display becomes enabled again when the edited contents are determined.

(3) Blank lines, which are inserted automatically to align the left- and right-side programs for the synchronization position, cannot be pasted with the menu [Paste] operation when the blank lines are specified as the copying range with the menu [Copy] operation.

(4) The editable machining program size is up to 240 KB per part system.

- When the menu [Synchro view] is pressed in the state where a machining program exceeding 240 KB is opened, the "Can't edit because of size over" message is displayed and the synchronous display cannot be performed.
- When a file exceeding 240 KB is opened with [Synchro view] highlighted, the "Can't edit because of size over" operation message is displayed and the machining program cannot be opened.

(5) The state where the [Synchro view] is highlight is held until the power is turned OFF and ON, a machining program for which the synchronous display is not available is opened, or the display type is changed to other than "Multi-program display type".

(6) When the G code guidance, etc. is opened in the 3-program/4-program simultaneous display, the other two program edit areas show timing synchronization of the part systems.

(7) If a timing synchronization symbol has no counterpart, the message "Can't display synchronous program" appears and the subsequent synchronous display is not performed.

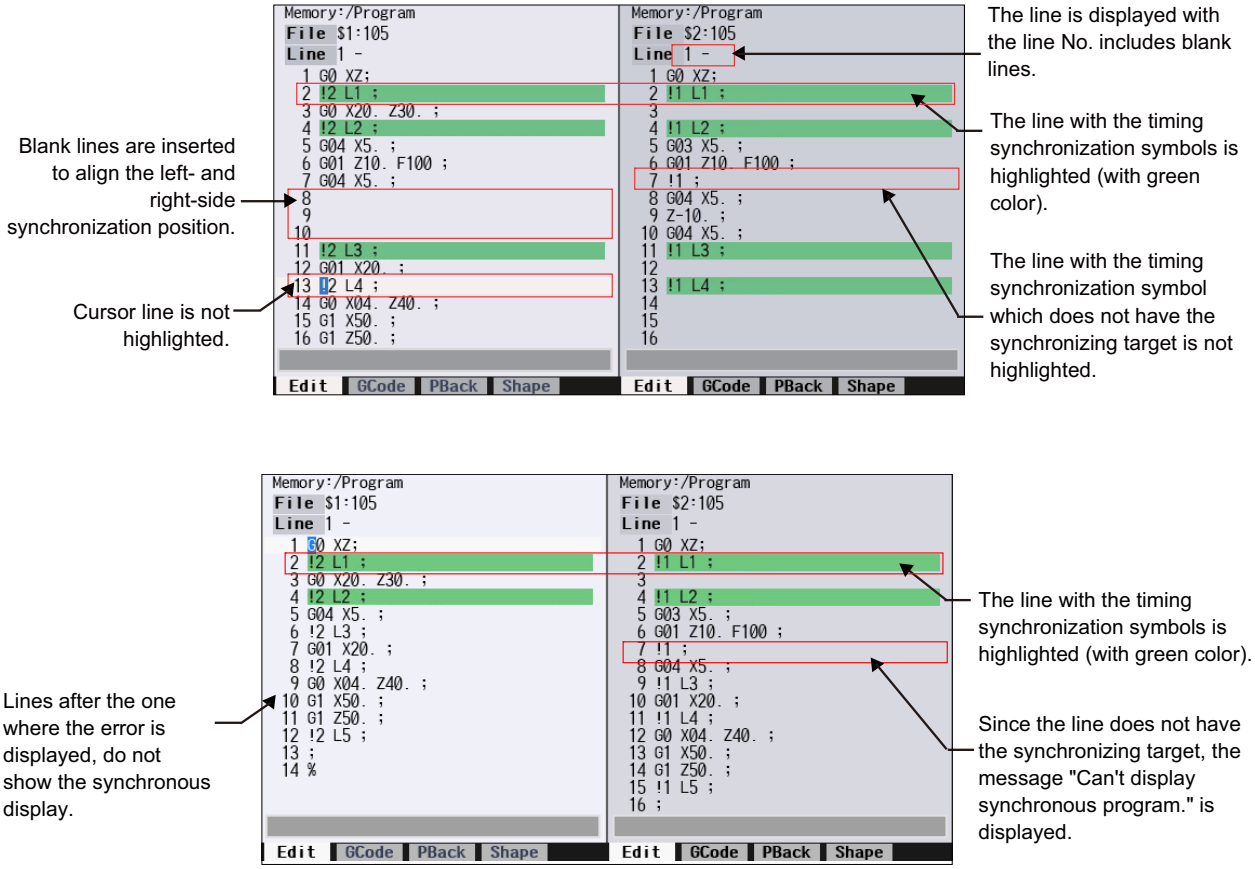
(8) Even when M code macro or M command synchronous tapping is overlapped with M codes specified by the parameters (#1310 and #1311) or the timing synchronization symbol definition files for synchronous view, the screen is displayed synchronously.

(9) If the synchronization M code specified by the parameter is not described in a single block, it is not treated as timing synchronization symbol.

(10) Even when the parameters (#19419, #1310 or #1311) are changed while the menu [Synchro view] is highlighted, re-synchronization is not performed. To reflect the change of the parameters, the menu [Synchro view] needs to be highlighted again.

(11) On the synchronous display with the menu [Synchro view], "L" (synchronization No.) which follows the synchronization symbol "!" cannot be displayed. Thus, the different "L" (synchronization No.) may be displayed synchronously on the 4-program simultaneous display screen.

[Display during synchronous view]

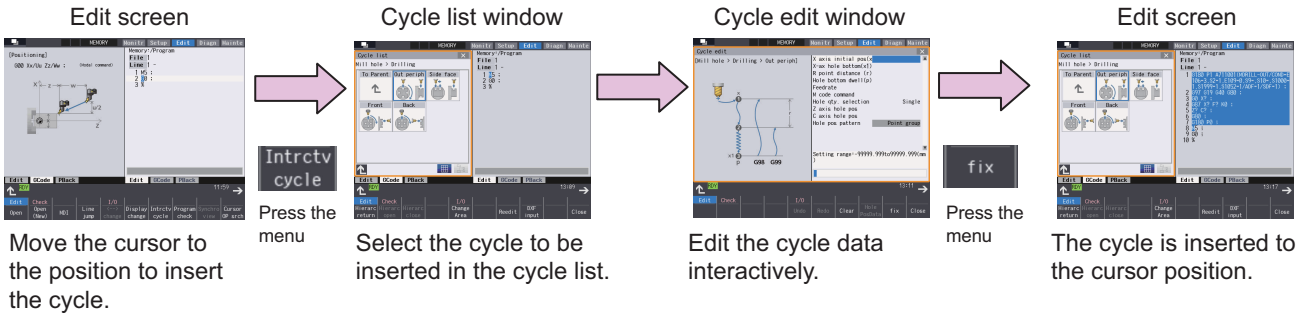


3.9 Inserting Cycle Interactively

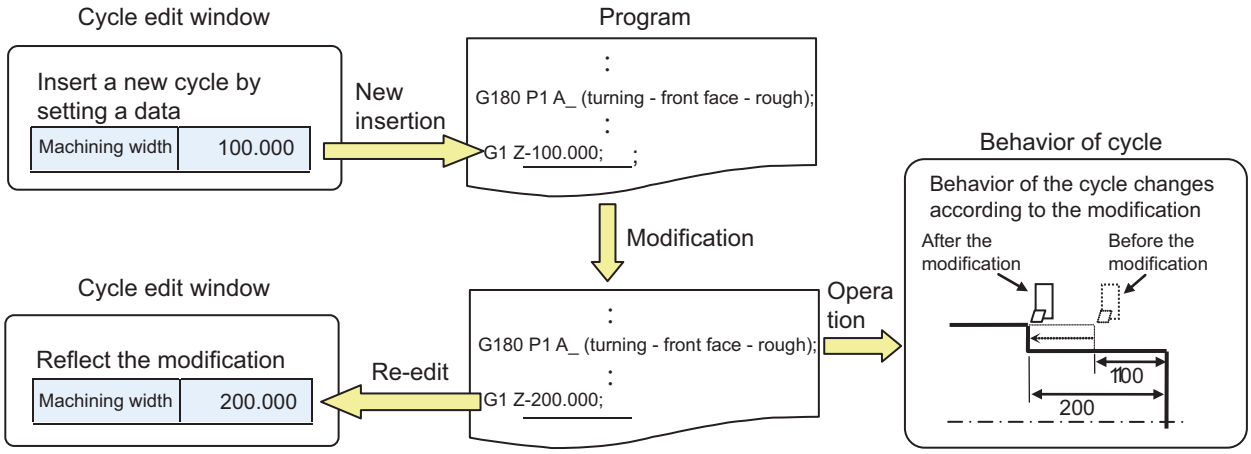
Edit – Edit

This function enables you to interactively insert a cycle that assists machining and setup for the program opening on the edit screen.
The programming time can be reduced by using this function.
The cycle can be easily inserted by editing the data on the interactive window.

[Operation image of cycle insertion]



Once you insert a cycle block, you can edit it directly on the program edit screen, or re-edit it in the cycle edit window if the cycle format is not changed.



3.9.1 Types of Cycle

Machining cycle can be inserted.

Cycle for lathe system or cycle for machining center system can be selected according to the NC system set with the parameter "#1007 System type select".

For cycle types, there are "Normal type" and "Extend type" of MITSUBISHI cycle and arbitrary cycle "Custom type" prepared by MTB.

Select the menu [Normal], [Extend] or [Custom] on the interactive cycle insertion screen. Select a cycle type you want to change and switch the cycle on the displayed cycle list window.

■Cycles for lathe system

○: Cycle insertion available -: No specification

Hierarchy 1	Hierarchy 2	Hierarchy 3	Standard type	Expansion type
Cycle type	Machining cycle	Machining portion		
Turning hole	Drill	Front	○	○
	Deep-hole drilling	Back	○	○
	Tapping		○	○
	Deep-hole tapping		○	○
	Reaming		-	○
	Boring		○	○
Turning	Turning	Outer diameter	○	○
	Turning (compound)	Inner diameter	○	○
	Copying	Front	○	○
	Grooving	Back	○	○
	Cutting-off		○	○
	Thread cutting	Outer diameter	○	○
	Thread cutting (compound)	Inner diameter	○	○
	Thread recutting		○	○
	Thread recutting (compound)		○	○
Milling hole	Drill	Periphery	○	○
	Deep-hole drilling	Side	○	○
	Step drilling	Front	○	○
	Tapping	Back	○	○
	Deep-hole tapping		○	○
	Reaming		-	○
	Boring		○	○
Milling	Pocketing	Side	-	○
	Grooving	Front	-	○
		Back		

Note

(1) For the standard type, there are two types of formats, normal format and MITSUBISHI CNC special format. It depends on the setting of the parameter "#1265 ext01" bit0 to bit2 which format is to be selected.

(2) The fixed cycle is used for "Tapping" and "Deep-hole tapping" for the expansion type. Same as the standard type, it depends on the setting of the parameter "#1265 ext01/bit2" which format is to be selected.

■**Cycles for machining center system**

○: Cycle insertion available -: No specification

Hierarchy 1	Hierarchy 2	Standard type	Expansion type
Cycle type	**Machining cycle**		
Hole	Drill	○	○
	Deep-hole drilling	○	○
	Step drilling	○	○
	Tapping	○	○
	Deep-hole tapping	○	○
	Reaming	-	○
	Boring	○	○
Milling	Plane	-	○
	Pocketing	-	○
	Grooving	-	○
	Contour	-	○

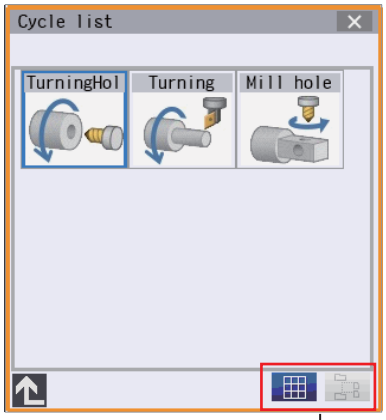
Note

(1) The fixed cycle is used for "Tapping" and "Deep-hole tapping" for the expansion type.

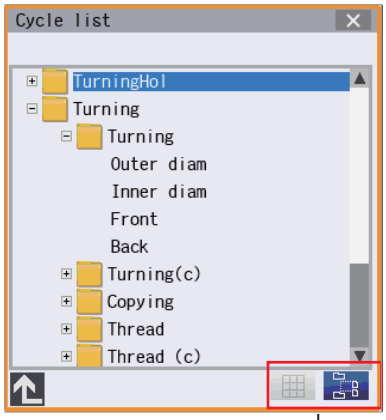
■**Display of cycle list window**

The cycle list window is displayed in the area opposite to editing area. For the cycle list window, there are "Image display type" and "Tree display type", which can be switched with change list button.

Image display type



Tree display type



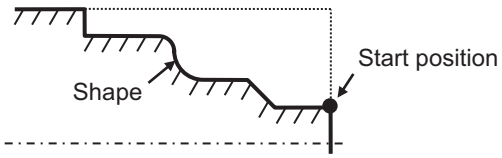
Change list button

3.9.2 Name of Data

Names and meanings of data related to cycle are indicated as follows:

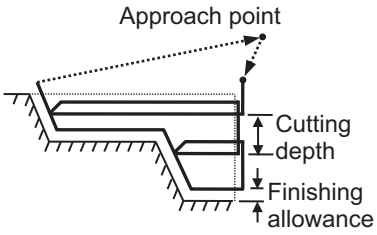
■Arbitrary shape data: Data to set the arbitrary shape

<Example of turning - outer diameter>

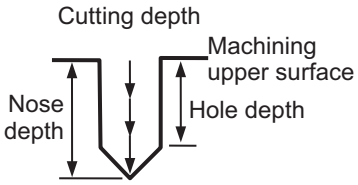


■Machining data: Data to specify the machining method as cutting depth and finishing allowance

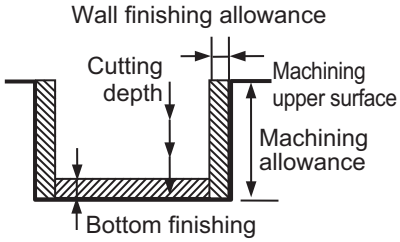
<Example of turning cycle>



<Example of drilling cycle>

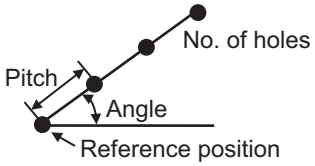


<Example of milling cycle>

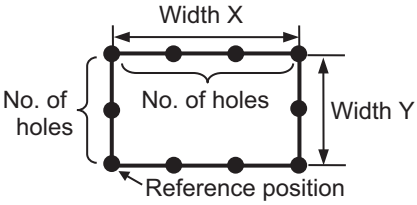


■Hole position data: When multi-drilling is performed, this is the data to specify the hole position.

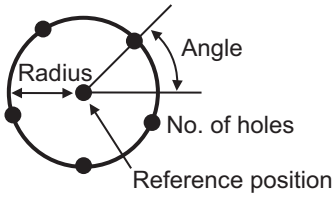
<Example of straight line hole position>



<Example of rectangle hole position>



<Example of circle hole position>



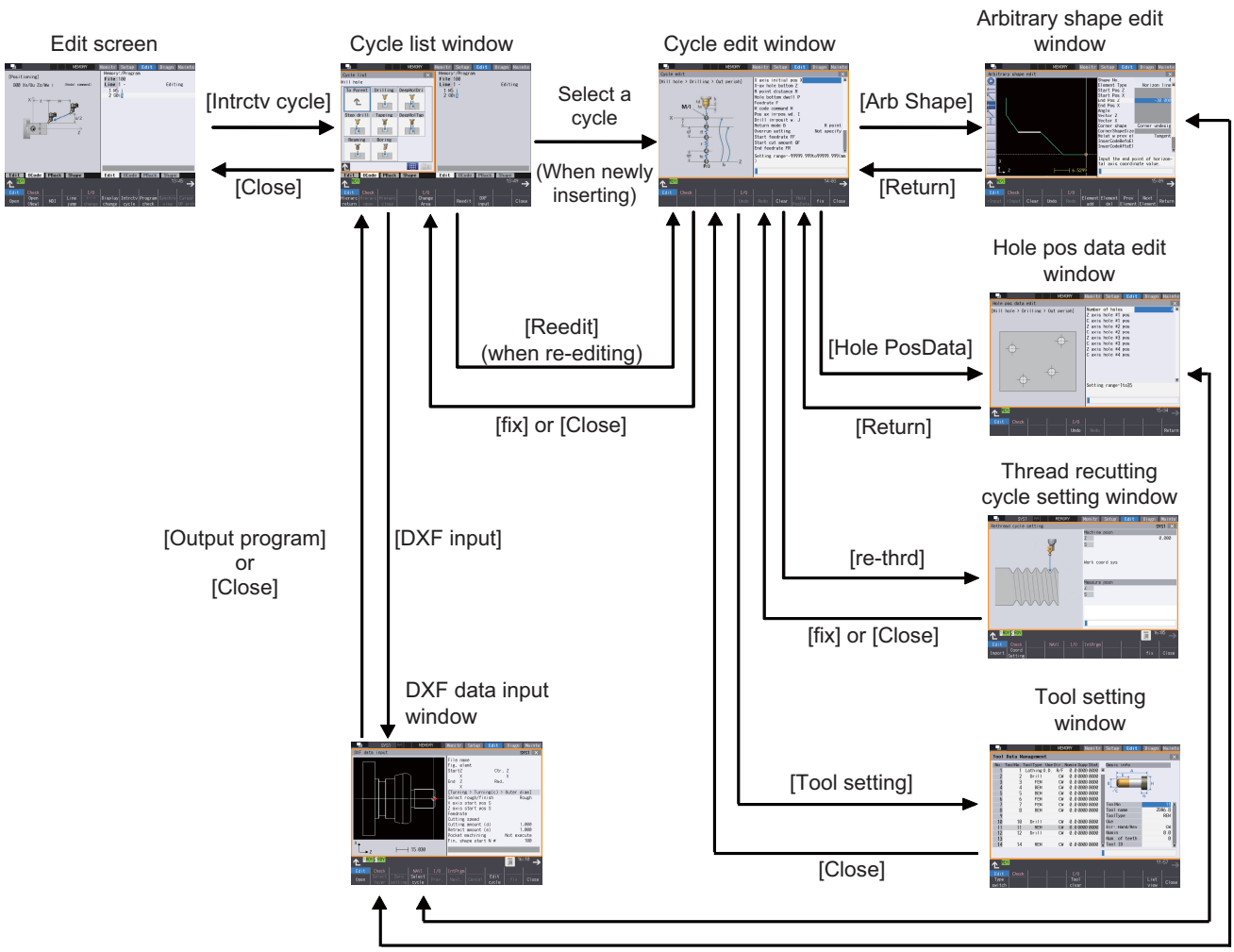
There are the following combinations for the data to set depending on the cycle types.

- Machining data only
- Machining data + Arbitrary shape data
- Machining data + Hole position data

Refer to the section "3.9.3 Screen Configuration" for the data edit window.

3.9.3 Screen Configuration

Screen configuration of this function is as follows:



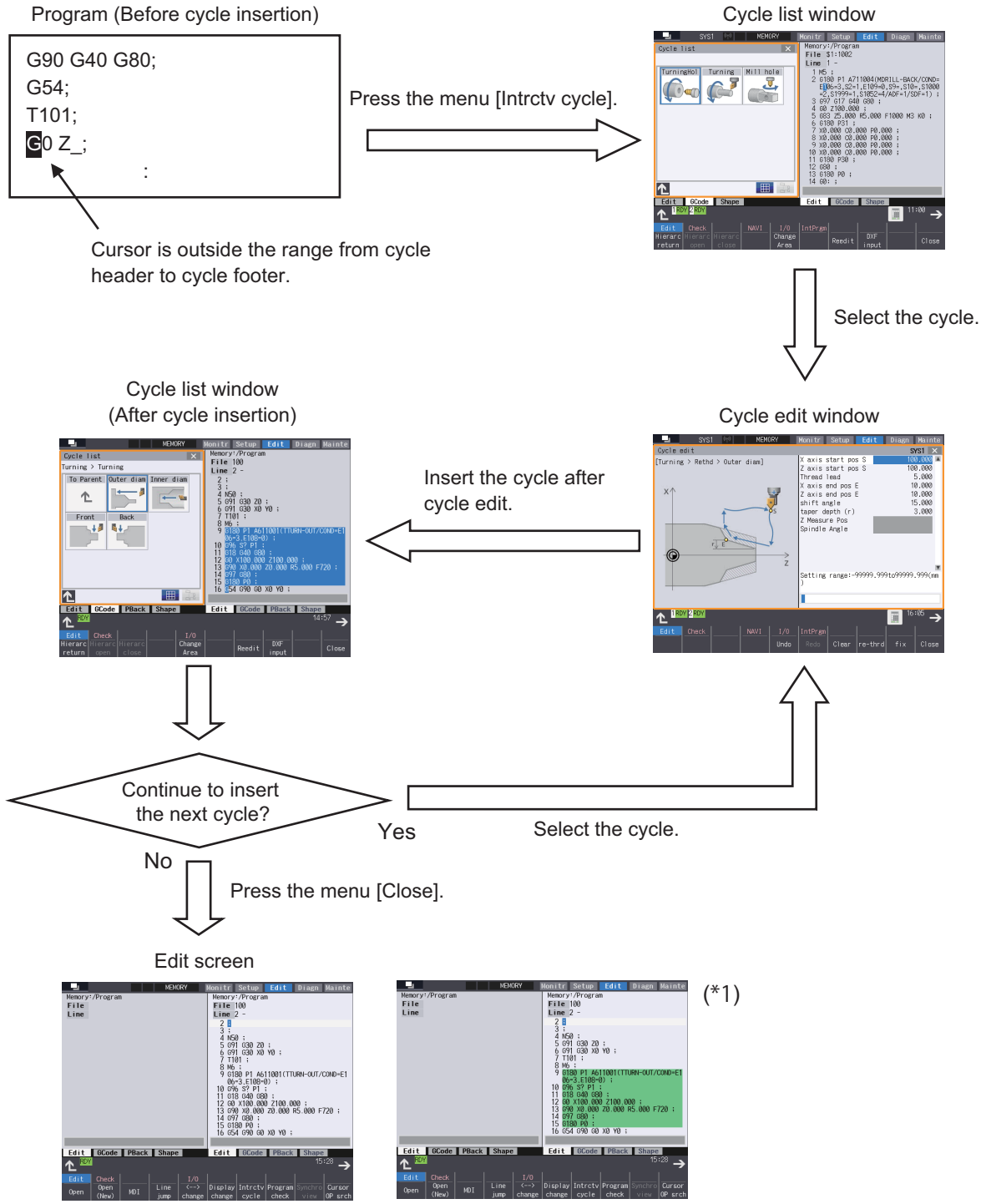
Screen/window	Details
Edit screen	This is a screen to edit the program. To insert or re-edit the cycle, press the menu [Intrctv cycle] on this screen.
Cycle list window	This is a window to select the cycle to be inserted.
Cycle edit window	This is a window to edit machining data of the cycle.
Arbitrary shape edit window	This is a window to edit arbitrary shape data of the cycle.
Hole pos data edit window	This is a window to edit hole position data of the cycle.
Thread recutting cycle setting window	When you are going to perform thread recutting for a machining completed workpiece, this is the window to read the measured thread groove position.
DXF data input window (*1)	This is a window to edit arbitrary shape data/hole position data of the cycle using DXF drawing file.
Tool management window	This is a window to set and display the tool information to be used for machining.

(*1) DXF data input function is a support for the interactive cycle insertion.
The function allows you to import a DXF drawing file from an external I/O device through DXF data input window and extract the figure element data from the drawing. And add them as arbitrary shape data or as hole position data to the cycle being edited using the interactive cycle insertion function.

3.9.4 Operation Descriptions

Inserting a new cycle program

In the program being opened on the edit screen, move the cursor to the position where you want to insert new cycle. Press the menu [Intrctv cycle] to select the cycle, and edit and insert it. The cycle is inserted before the block where the cursor is located in the program being opened on the edit screen.

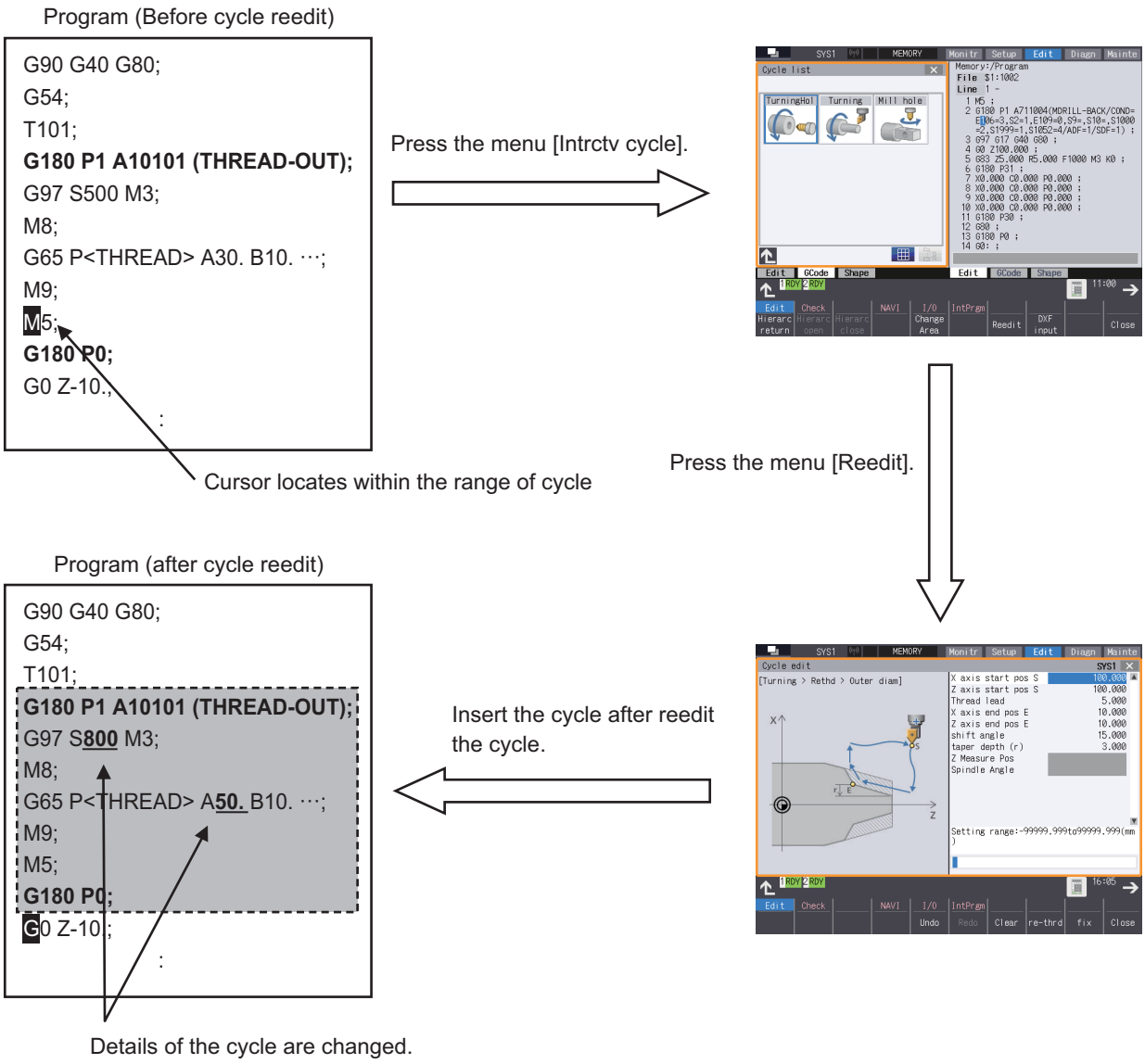


After the cycle insertion, the screen returns to the cycle list window in the cycle inserted state. The inserted cycle is highlighted in the editing area of the program, and the cursor moves to the head of the next line of the cycle. Then, the program is saved.

(*1) The block from cycle header to cycle footer can be highlighted with the parameter setting.
For details, refer to "3.9.6 Highlighting the Cycle Program".

Reediting the cycle program

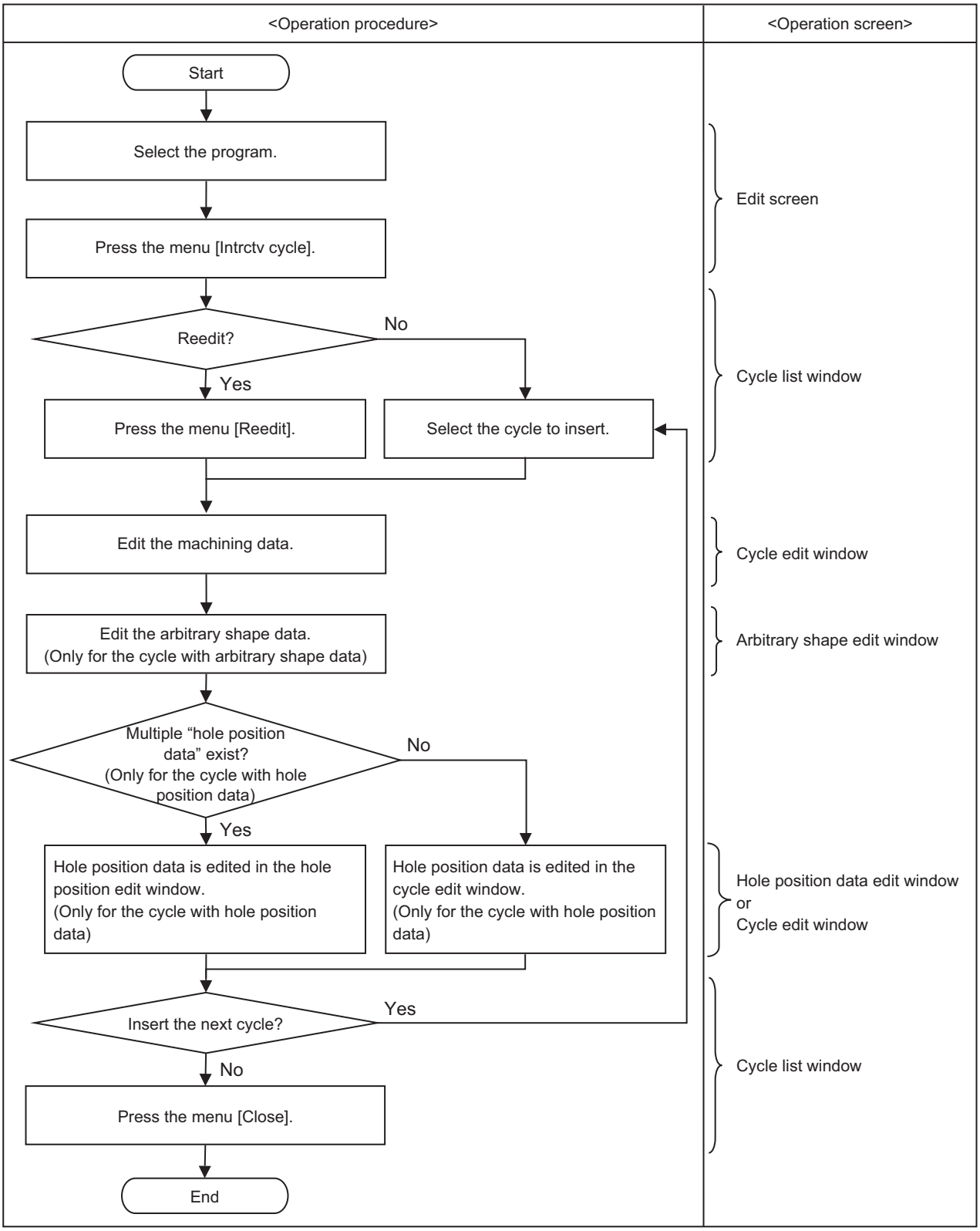
In the program being opened on the edit screen, move the cursor within the range from cycle header (G180P1 block) to cycle footer (G180P0 block) of the cycle that you want to reedit. Press the menu [Intrctv cycle] and then [reedit] in this condition, existing cycle can be reedited.



After reediting the cycle, the screen returns to the cycle list window in the state that the reedit results were reflected to the cycle. The inserted cycle is highlighted in the editing area of the program, and the cursor moves to the head of the next line of the cycle. Then, the program is saved.

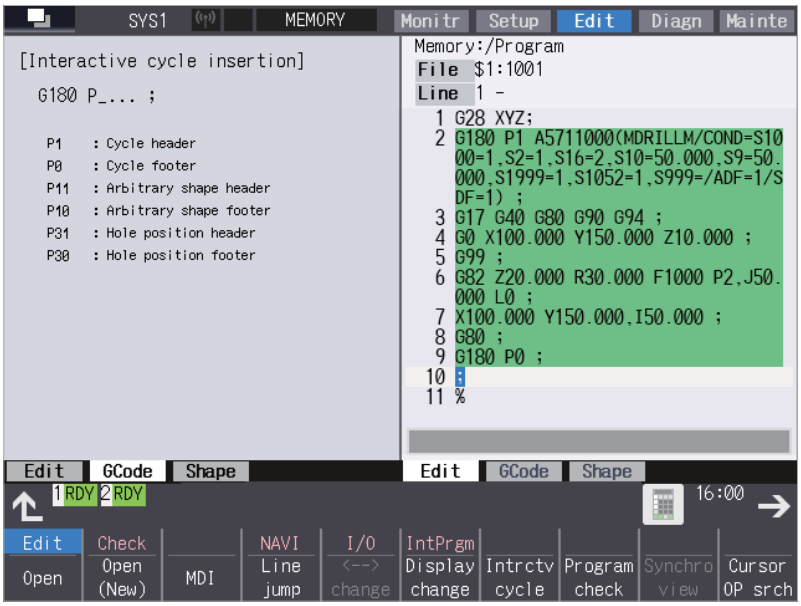
3.9.5 Operation Procedure

The chart below indicates the operation procedure of cycle insertion and reediting.



3.9.6 Highlighting the Cycle Program

When the parameter "#8993 Cycle highlight" is set to "1", the blocks from cycle header (G180P1) to cycle footer (G180P0) are highlighted. The block where the cycle is inserted can be distinguished easily by using this function.



Note

(1) When the set of cycle header and cycle footer does not exist within 200 blocks above and below from the first line of the display area, the block is not highlighted.

(2) The highlighted display is not updated while the program is being edited.

(3) The highlighting function does not perform the format check of the cycle. Therefore, even in the highlighted block, it may not be possible to re-edit the cycle.

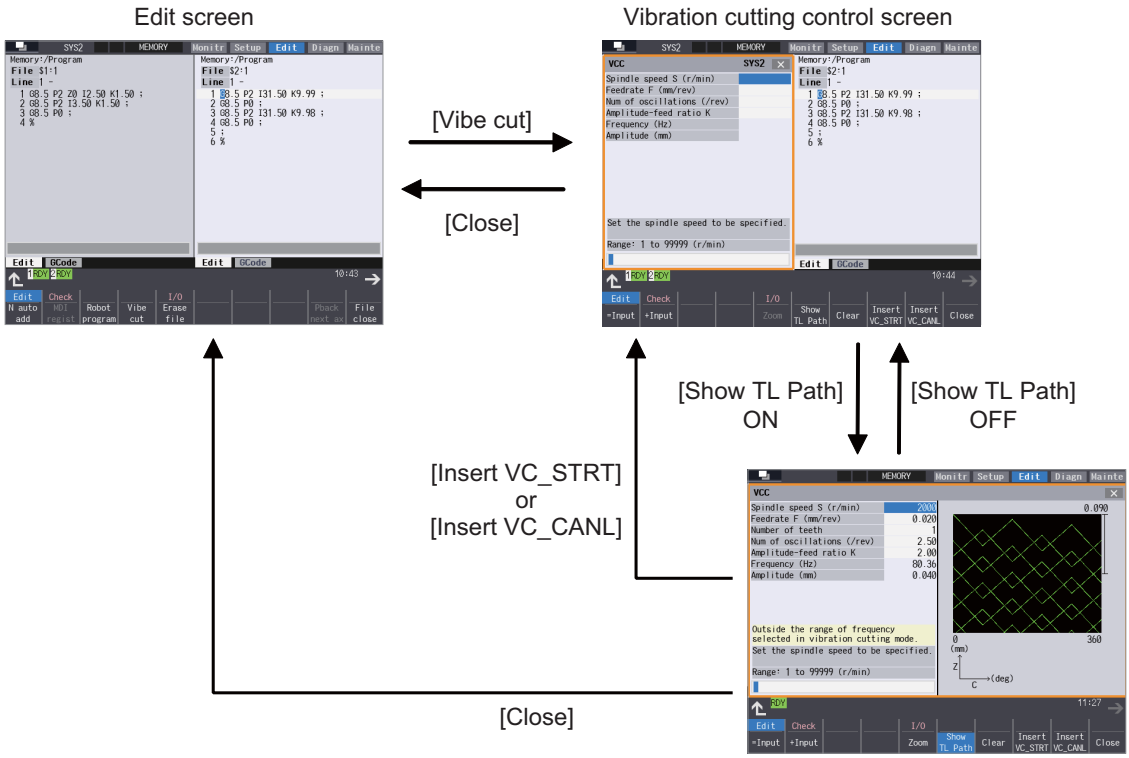
3.10 Setting Machining Conditions for Vibration Cutting Control

Edit – 

On the vibration cutting control screen, theoretical tool paths are drawn according to the data by entering the spindle rotation speed, feedrate, and the number of oscillations specified with the machining program G8.5 command.
The vibration conditions and tool paths displayed on this screen are theoretical values. The actual tool paths depend on the machine.

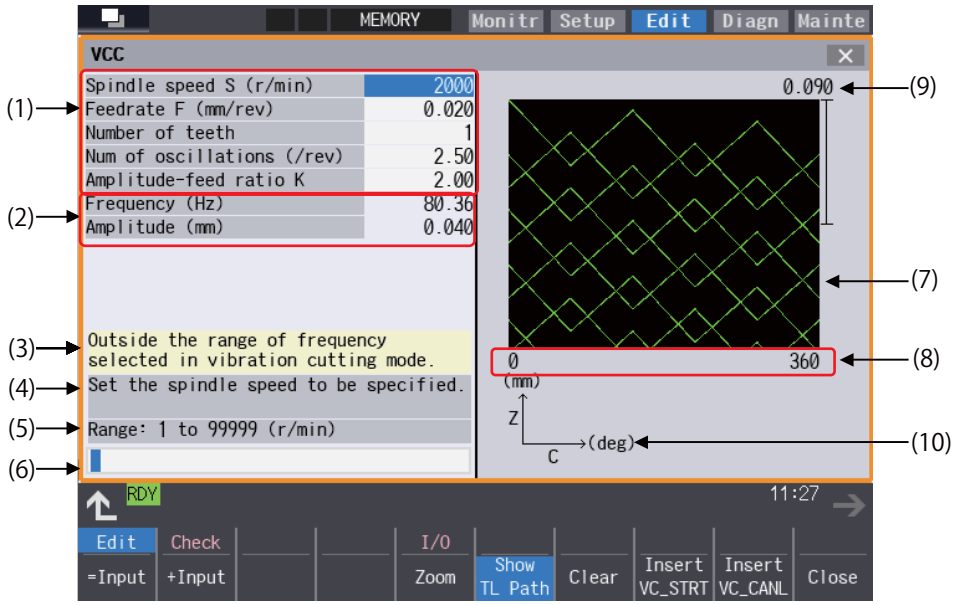
3.10.1 Screen Configuration

Screen configuration of this function is as follows:



Screen	Details
Edit screen	This is a screen to edit the program. Press the menu [Vibe cut] to display the vibration cutting control screen (without tool path diagram).
Vibration cutting control screen (without tool path diagram)	The machining conditions can be set by referring to the program on the edit screen.
Vibration cutting control screen (with tool path diagram)	The machining conditions can be set by referring to the tool path diagram.

The following is the vibration cutting control screen (with tool path diagram) for M system.



Display items

Display items		Details
(1)	Machining condition setting display area	The vibration cutting conditions are displayed/set.
(2)	Vibration condition display area	The theoretical value of the vibration conditions is displayed.
(3)	Warning message	A warning message is displayed when the frequency selected in the vibration cutting mode is out of the range. The upper limit and lower limit of the frequency are determined depending on the setting values of the parameters "#11678 VibPerRevMax_VCC", "#12131 FrqClmpSys_VCC" and "#11686 Spec_VCC/bit5".

(1) Machining condition setting display area

Display	Explanation
Spindle speed S (r/min)	Set the spindle rotation speed to be commanded.
<Metric> Feedrate F (mm/rev) <Inch> Feedrate F (inch/rev)	Set the tool feedrate to be commanded.
Number of teeth	Set the number of teeth of the tool. <Note> ♦This is not displayed for L system.
Num of oscillations (/rev)	Set the number of oscillations per one rotation of the spindle in the vibration cutting mode.
Amplitude-feed ratio K	Set the amplitude feed ratio in the vibration cutting mode.

(2) Vibration condition display area

Display	Explanation
Frequency (Hz)	The theoretical value of the frequency is displayed.
<Metric> Amplitude (mm) <Inch> Amplitude (inch)	The theoretical value of the amplitude is displayed. <Metric> The fourth decimal place is rounded down. <Inch> The fifth decimal place is rounded down.

Display items		Details
(4)	Guidance display	The guidance of the item at the cursor position is displayed.
(5)	Setting range display	The setting range of the item at the cursor position is displayed.
(6)	Input area	The input contents are displayed.
(7)	Tool path diagram	The theoretical value of the tool path is displayed in a diagram. The theoretical value of the tool path can be compensated according to the parameter "#11687 AmpDspCorrRate_VCC". The amplitude feed ratio to be used to calculate the theoretical value of the tool path is a value obtained by multiplying "Amplitude-feed ratio K" entered in the machining condition setting area by the magnification set in #11687. It is displayed in the vibration waveform selected in the parameter "#11673 VibType_VCC". <Note> ♦When all the items in the machining condition setting display area are entered, the tool path is displayed. When any item is not entered, the tool path is not displayed.
(8)	Rotation angle display	The drawing range of the spindle rotation angle is displayed. (The first decimal place is rounded down.) The unit is (°).
(9)	Scale display	The scale of the drawing area is displayed. The unit is set to (mm/100pixel) for metric system and (inch/100pixel) for inch system.
(10)	Coordinate system display	The coordinate system direction of the tool path diagram is displayed. ♦The Z axis in the coordinate system indicates the feed axis position. However, the Z axis direction is opposite to the cutting direction. ♦The C axis in the coordinate system indicates the spindle rotation position (degree). The Z and C axes are always displayed in the coordinate system regardless of the axis setting. <M system> The Z axis indicates the upward direction, and the C axis indicates the rightward direction. <L system> The Z axis indicates the rightward direction, and the C axis indicates the downward direction.

Menus

Menus	Details
=Input	This executes an absolute input.
+Input	This executes an incremental input.
Zoom	This switches the normal display mode and the zoom-in mode.
Show TL Path	This switches the program display and the tool path diagram.
Clear	This erases the setting value at the cursor position.
Insert VC_STRT	This inserts the vibration cutting mode start command into the block before the cursor position in the program.
Insert VC_CANL	This inserts the vibration cutting mode cancel command into the block before the cursor position in the program.
Close	This returns the screen from the vibration cutting control screen to the edit screen.

3.10.2 Operation Method

The following describes the operation procedure to insert the vibration cutting control ON/OFF command into the program.

Displaying the tool path diagram

The following describes the procedure to adjust the command at the start of the vibration cutting mode, with the program example.

[Example of machining program]

```
      G53 X0. Z0. ;
      M101 ;                    ..... 1st spindle selection
      G95 ;                     ..... Feed per revolution (synchronous feed)

(a)   M3 S2000 ;                ..... Spindle forward rotation
                                       Spindle rotation speed: 2000 (r/min)
(b)   G8.5 P2 I2.5 K2.0 ;       ..... Start of vibration cutting mode
                                       Spindle rotation speed: 1928 (r/min)
      G00 X50. Z80.
      N01 G00 X20. ;

(c)   N02 G01 Z45. F0.02 ;      ..... Vibration cutting control (Z axis) is enabled.
      N03 X35. Z25. F0.04 ;     ..... Vibration cutting control is disabled.
      N04 X45. F0.02 ;          ..... Vibration cutting control (X axis) is enabled.

      N05 G00 X50. ;
      N06 Z80. ;

      G8.5 P0 ;                 ..... Cancel of vibration cutting control
                                       Spindle rotation speed: 2000 (r/min)
      M5 ;                      ..... Spindle stop
                                       Spindle rotation speed: 0 (r/min)

      M30 ;
```

(a) Spindle rotation speed S (b) Number of oscillations I, Amplitude-feed ratio K (c) Feedrate F

(1) Press the menu [Open] on the edit screen to open the machining program.

(2) Move the cursor to the start position of vibration cutting.

(Example)

Move the cursor to "G" at the head of (b) in the machining program example.

(3) Press the menu [Vibe cut].

(4) Enter the spindle rotation speed to be commanded, and press the [INPUT] key or the menu [=Input]. → The value is set.

(Example)

According to (a) in the machining program example
Spindle rotation speed S: 2000

It is possible to perform incremental input by pressing the menu [+Input] key instead of the [INPUT] key or the menu [=Input].

(Example) When the original value is "200"

250 [+Input] -> 200 + 250 = 450

(5) Enter the other items in the same way.

(Example)

According to (c) in the machining program example
Feedrate F: 0.02

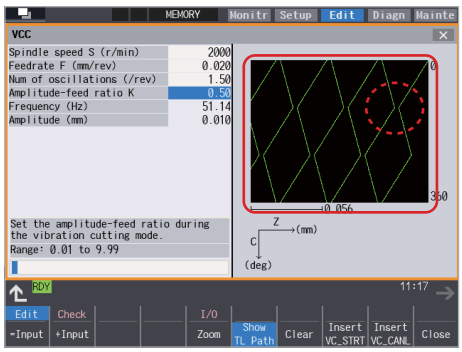
Set values within the setting range for "Number of oscillations I" and "Amplitude-feed ratio K".

(Example)

Number of oscillations I: 1.5
Amplitude-feed ratio K: 0.5

(6) Press the menu [Show TL Path]. → The tool path diagram is displayed.

In this example, the tool is not separated from the workpiece; therefore, there is no air-cutting zone, and the chips are not broken up.



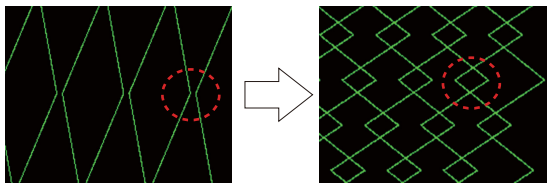
(7) Adjust the "Number of oscillations I" and "Amplitude-feed ratio K" so that air-cutting zone is generated.

(Example)

Number of oscillations I: 2.5
Amplitude-feed ratio K: 2.0

The tool path diagram is updated.

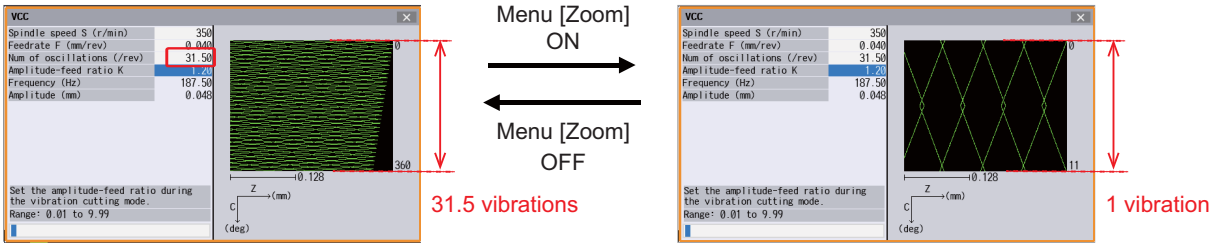
With this setting, air-cutting zone is generated to separate the tool from the workpiece, and the chips are broken up.



Insert the adjusted values into the program with the following procedure.

■Zooming in on the tool path diagram

When the value of "Num of oscillations" is too large to see the tool path diagram, press the menu [Zoom] to zoom in on the path diagram to show one vibration.

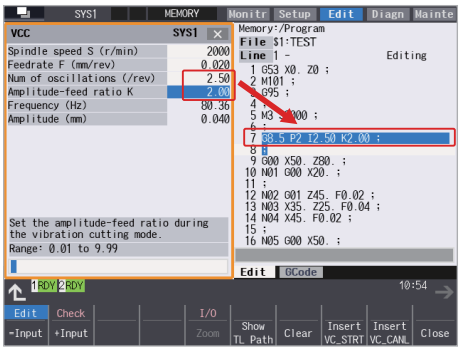


Inserting the vibration cutting mode start command into the program

(1) Confirm that the cursor is at the position to insert the vibration cutting mode start command, and press the menu [Insert VC_STRT].

The vibration cutting mode start command is inserted at the cursor position of the machining program.

When the tool path diagram is displayed, the display changes to the program display.
The machining condition setting is applied to the inserted vibration cutting mode start command.



Inserting the vibration cutting mode cancel command into the program

(1) Move the cursor to the position to insert the vibration cutting mode cancel command, and press the menu [Insert VC_CANL].

The vibration cutting mode cancel command is inserted at the cursor position of the machining program.

When the tool path diagram is displayed, the display changes to the program display.

3.11 Process Analysis Function

Edit – 

(For 15-type/19-type display unit)

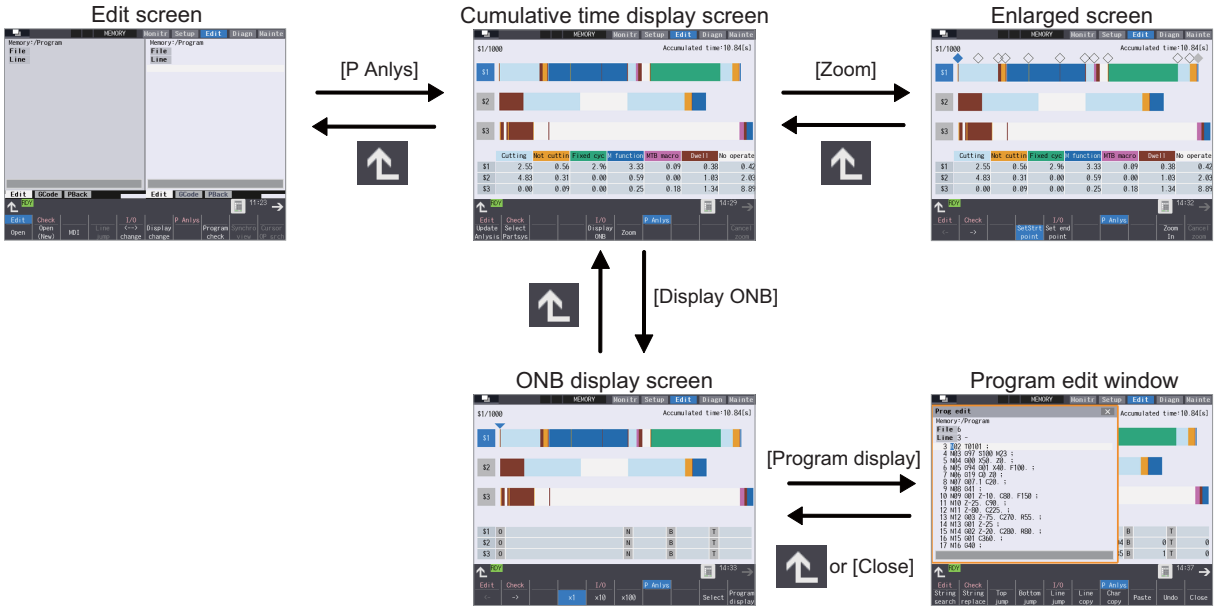
Edit – 

(For 8.4-type/10.4-type display unit)

The process analysis function enables to store execution information of the machining program and display the analysis results of the data on the screen. When the program is optimized, this function supports to find a point to reduce the cycle time. A program with up to approximately 30000 blocks in total can be analyzed.

3.11.1 Screen Configuration

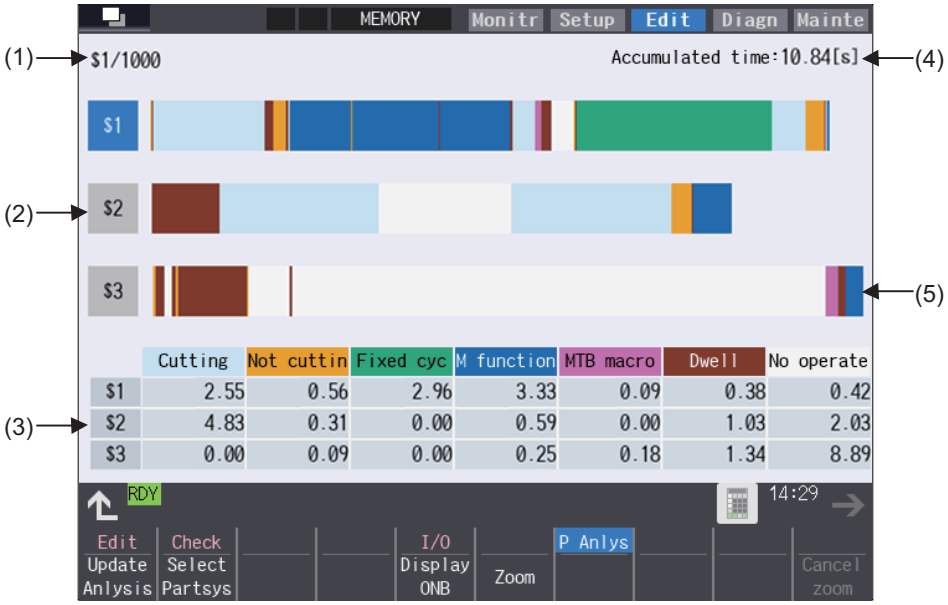
Screen configuration of this function is as follows:



Screen	Details
Edit screen	This is a screen to edit the machining program. To display the cumulative time display screen, press the menu [P Anlys] on this screen.
Cumulative time display screen	This is a screen to display the analysis results of the machining program execution information in a graph.
ONB display screen	This is a screen to select a position in the graph and display the O/N/B/T numbers of the selected position.
Enlarged screen	This is a screen to enlarge the area selected in the graph.
Program edit window	This is a window to edit the machining program.

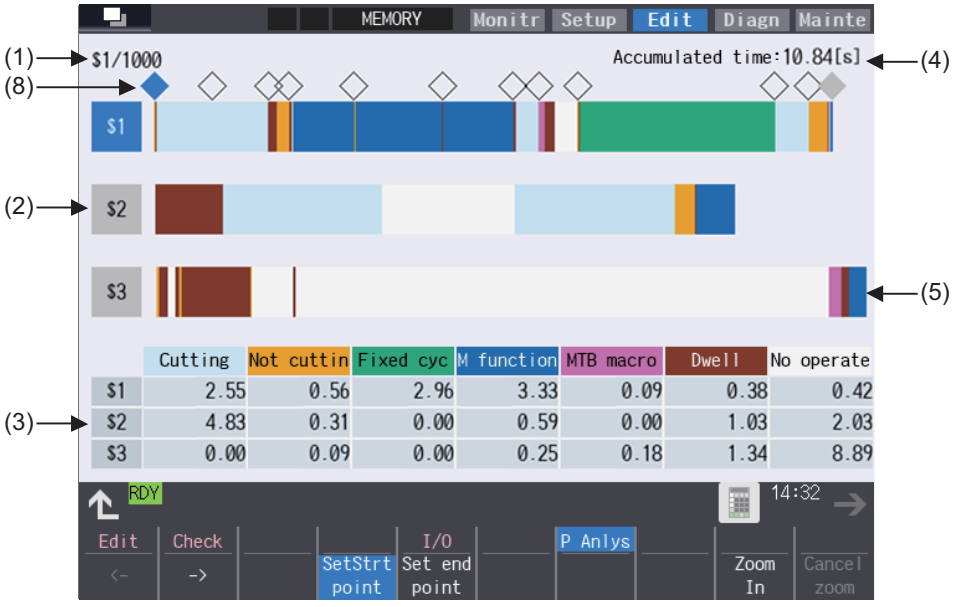
■Cumulative time display screen

When the menu [P Anlys] is pressed on the edit screen after the machining program for process analysis is executed, the analysis results of the machining program execution information are displayed in a graph.
The state of commands such as cutting, non-cutting, miscellaneous functions, etc. in the machining program, and the time taken for each command process can be checked.



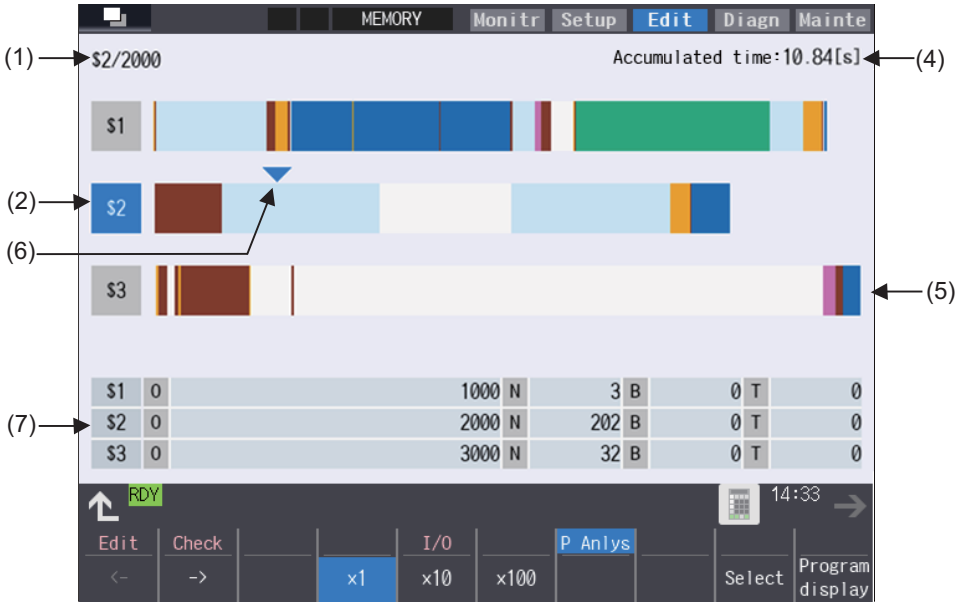
■Enlarged screen

When the menu [Zoom] is pressed on the cumulative time display screen, the enlarged screen is displayed. On this screen, the area selected in the graph can be enlarged and the analysis results can be checked in more detail. The selected area is maintained even if the screen switches to another screen.



■ONB display screen

When a position is selected in the graph displayed on the cumulative time display screen and the menu [Display ONB] is pressed, the O/N/B/T numbers (*1) of the selected position are displayed.



(*1) The O/N/B/T numbers indicate the following numbers.

O: Program No.

N: Sequence No.

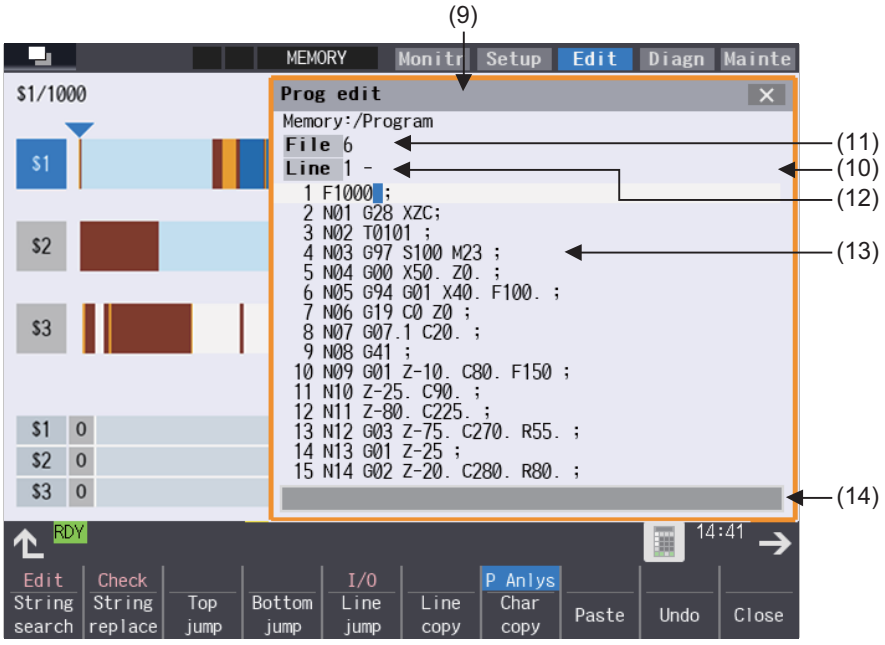
B: Block No.

T: Tool No.

■Program edit window

When the menu [Program display] is pressed on the ONB display screen, the machining program at the selected position is opened. The contents can be checked from the NB number position of the machining program. Both regular editing and mass-editing are available. For the conditions of mass-editing, refer to "3.2 Creating and Editing a Machining Program".

<Regular editing>



Display items

	Display items	Details
(1)	Selected part system program name	The program name of the part system selected in (2) is displayed. It is displayed after the process analysis is completed.
(2)	Part system (*1)	The part system displayed for the graph is displayed. Select the part system to display by the menu [Select Partsys] on the process analysis screen (cumulative time display). Up to three part systems can be displayed. The selected part system is switched by touch gestures or the cursor keys [↑] and [↓].
(3)	Cumulative time display for each type (*1)(*2)	The cumulative time for each block type is displayed. The displayed part system is linked with the part system of (2).
(4)	Cumulative time display	Maximum cumulative time of all the valid part systems is displayed. This item is displayed after the process analysis is completed.
(5)	Graph display	The analysis results are displayed in a graph. The graph is displayed based on the maximum cumulative time. (*4) The seven block types are displayed: "Cutting", "Not cuttin", "Fixed cyc", "M function", "MTB macro", "Dwel" and "No operate". It is displayed after the process analysis is completed.
(6)	Selection cursor	The selection cursor is displayed at the top of the graph of the part system selected in (2). The information (O/N/B/T numbers) of the position specified by the selection cursor is displayed in the machining program display of (7). The selection cursor can be moved by touch gestures on the graph, menu selection and key input.
(7)	Machining program display (*1)(*3)	The machining program (O/N/B/T numbers) of the selection cursor position is displayed. The O/N/B/T numbers are updated to the information of the selection cursor position by pressing the menu [Select]. <Note> ♦For the deselected part system, since the machining program for the same time as the selected part system is displayed, the O/N/B/T numbers might be different from the type displayed in the graph.
(8)	Position to enlarge the area (*5)(*6)	Candidate points for the area to be enlarged are displayed at the top of the graph of the part system selected in (2). Select the start and end points from the candidate points and press the menu [Zoom In] to display the graph of the enlarged area from the start point to the end point. The enlarged graphs of the same area are displayed for the other part systems.
(9)	Path display	The path of the currently opened program file is displayed. (Example) Memory:/program When the path is too long, the portion exceeding the display area is not displayed.
(10)	Edit type display/Current edit display/Insert mode display	<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"> The overwrite mode and insert mode are switched by pressing the [INSERT] key. This is displayed when the insert mode is selected.
(11)	File name display	This displays the file name of the program currently being edited.
(12)	Top line of the displayed program/Display position	This displays the top line of the displayed program. During mass-editing, the position of the currently displayed page is displayed in percentage (rate) when the entire size of the program is assumed to be 100%.
(13)	Line number/Program display	<Line number> This displays the last 3 digits of the program line number. When over two or more lines are displayed on the screen because one line is long, the line No. does not attach to the beginning of the next line. The line No. does not attach during mass-editing. <Program display> The contents of the program currently being edited are displayed. The line that the cursor is on is highlighted. The line is not emphasized during mass-editing.

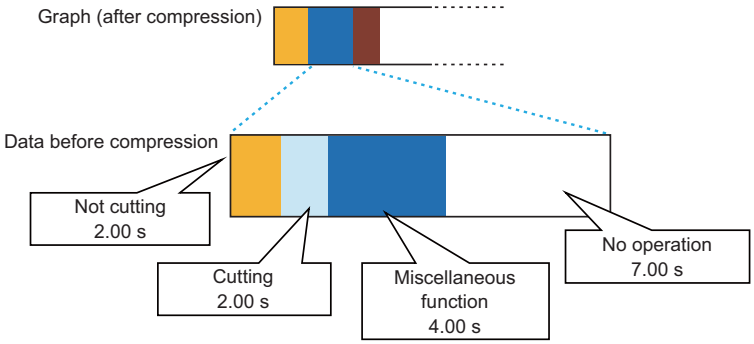
Display items	Details
(14) Input area	This displays details of the key input.

(*1) "$1" to "$8" are displayed as the part system names regardless of the setting of the parameters "#1169 System name" and "#12144 System name2".

(*2) When a value over 999999.99 is displayed, the numbers after the decimal point are omitted and the first integer digit is displayed with "*".
(Example) 1999999.99 -> 199999*

(*3) When the machining program of the selection cursor position is MTB macro or fixed cycle, only the program No. is displayed. A blank is displayed for other items.

(*4) The following is an example of graph compression display.



In the above figure, the type whose cumulative time is the longest before compression is "No operation".
However, "No operation" is not displayed when at least one of the other types is displayed.
Therefore, "Miscellaneous function" whose cumulative time is the second longest is the type to be displayed as a typical one after compression.
When there are two or more types whose cumulative time is the longest, they are displayed in the following priority order.
Cutting -> Not cutting -> Fixed cycle -> Miscellaneous function -> MTB macro -> Dwel

(*5) Rhombuses are displayed at the candidate points.
There are the following rhombus types.


Neither start point nor end point


Start point or end point
The menu [SetStrt point] or [Set end point] is not selected


Start point or end point
The menu [SetStrt point] or [Set end point] is selected

(*6) The candidate points are displayed at both ends of the graph and at the positions where the machining program or sequence No. is changed.
The candidate points are decided excluding the commands such as macro statements, block with no movement, that are not displayed on the process analysis screen. Therefore, even when sequence Nos. are on the machining program, they may not be the candidate points.
A candidate point of which half or more overlaps the previous one is not displayed. In that case, it can be displayed by enlarging the area.

The following example shows candidate points.
This table does not include the right end candidate point of the graph.

Program	Sequence No.	Machining program example	Candidate point	Reason
100	1	N001 G0 X100.	○	It is a candidate point because it is the start point of the graph.
100	2	N002 G98 G1 X150. F1000	○	It is a candidate point because the sequence No. is changed.
100	2	M98 P200	×	It is not a candidate point because the machining program or sequence No. is not changed.
200	2	N002 G98 G1 X200. F1000	○	It is a candidate point because the machining program is changed.
200	2	M99	×	It is not a candidate point because the machining program or sequence No. is not changed.
100	3	N003 G4 P1000	○	It is a candidate point because the machining program and the sequence No. are changed.
100	3	M30	×	It is not a candidate point because the machining program or sequence No. is not changed.

Menus

Cumulative time display screen

Menus	Details
Update Anlysis	This starts analysis of the machining program execution information.
Select Partsys	This displays the sub-menu of part system selection to change the displayed part system.
Display ONB	This displays the ONB display screen.
Zoom	This displays the enlarged screen.
Cancel zoom	This cancels the enlarged state.

Sub-menus of [Select Partsys] menu

Menus	Details
$1	The selected part system is highlighted. Press the menu [Apply] to reflect it to the part system display.
$2	
$3	
$4	
$5	
$6	
$7	
$8	
Apply	The selected part system is applied to the screen.

ONB display screen

Menus	Details
<-	This moves the selection cursor to the left. Specify the cursor movement amount with the menu [x1], [x10] or [x100].
->	This moves the selection cursor to the right. Specify the cursor movement amount with the menu [x1], [x10] or [x100].
x1	This sets the movement amount of the selection cursor to the actual amount.
x10	This sets the movement amount of the selection cursor to the tenfold amount.
x100	This sets the movement amount of the selection cursor to the hundredfold amount.
Select	This displays the O/N/B/T numbers of the selection cursor position.
Program display	This displays the contents of the machining program corresponding to the selected cursor position from the NB numbers.

Enlarged screen

Menus	Details
<-	This moves the start point (or end point) of the area to be enlarged to the left.
->	This moves the start point (or end point) of the area to be enlarged to the right.
SetStrt point	When it is selected (highlighted), the start point can be moved.

Menus	Details
Set end point	When it is selected (highlighted), the end point can be moved.
Zoom In	This enlarges the area between the start point and the end point.
Cancel zoom	This cancels the enlarged state.

3.11.2 Operation Method

Displaying the analysis results

(1) Press the menu [P Anlys] on the edit screen after executing the machining program of process analysis target.
Press the menu [P Anlysis] for 15-type/19-type display unit.

(2) Press the menu [Update Anlysis].
The message "Process analysis in progress" is displayed in flickering until the analysis results are drawn.

<Note>

- The analysis results are not displayed when automatic operation is not executed or during automatic operation.

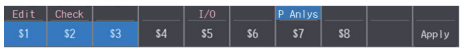
→ The analysis results are displayed when the process analysis is completed.



Changing the displayed part system

(1) Press the menu [Select Partsys] on the cumulative time display screen.

→ The sub-menu of part system selection is displayed. Only the valid part systems are displayed in the menu.

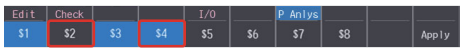


(2) Select the part system menu to switch between show and hide.
Up to three part systems can be selected.

<Example>

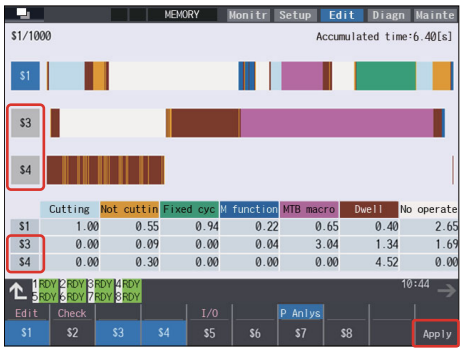
Change to hide the part system 2.
Change to show the part system 4.

→ The highlight of the part system 2 is canceled, and the part system 4 is highlighted.



(3) Press the menu [Apply].

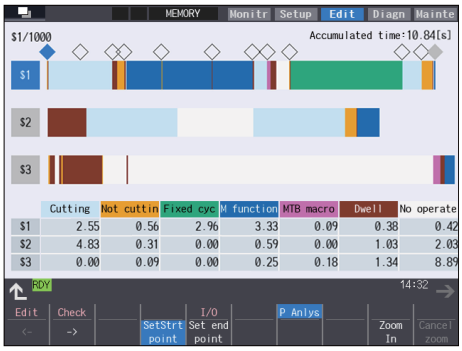
→ The screen is refreshed to display the selected part systems.



Zooming the graph

(1) Press the menu [Zoom] on the cumulative time display screen.

The enlarged screen is displayed.
The candidate points to specify the area to be enlarged are displayed at the top of the graph of the selected part system.



(2) Select the start point of the area to be enlarged. Press the menu [SetStrt point] and switch the start point by the menus [<-] and [->].

The selection cursor can also be moved by the following operation.

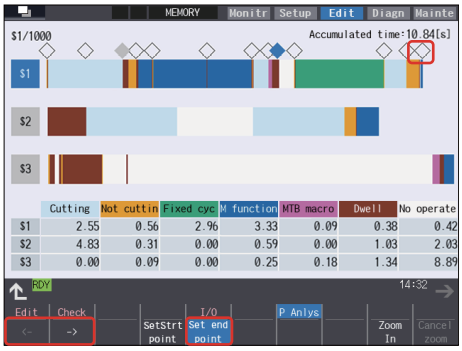
- Tap a candidate point on the graph.
- Use the cursor key [←] or [→] to move to any position.

When the menu [SetStrt point] is pressed, the current start point turns blue.
When the menu [<-] or [->] is pressed, the start point moves to the previous or next candidate point.
The start point can move only in the left area from the end point.



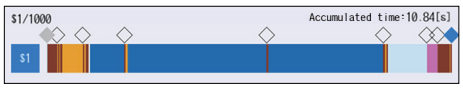
(3) Select the end point of the area to be enlarged. Press the menu [Set end point] and switch the end point by the menus [<-] and [->].

When the menu [Set end point] is pressed, the current end point turns blue.
When the menu [<-] or[->] is pressed, the end point moves to the previous or next candidate point.
The end point can move only in the light area from the start point.



(4) Press the menu [Zoom In] to determine the area to be enlarged.

→ The area from the start point to the end point is enlarged and displayed in a graph.
The cumulative time of the enlarged area is displayed in the cumulative time display for each type.



Checking the ONB numbers

(1) Press the menu [Display ONB] on the cumulative time display screen.

→ The machining program display at the bottom of the screen is changed from the cumulative time display to the ONBT number display.
The selection cursor is displayed at the top of the graph of the selected part system.

(2) Press the menu [<-] or [->] to move the selection cursor to a desired position.
The movement amount one time can be changed with the menu [x1], [x10] or [x100].

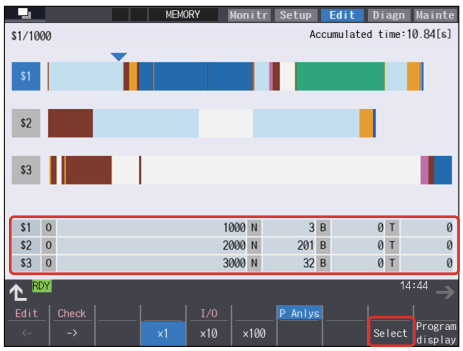
The selection cursor can also be moved by the following operation.

- Tap any position on the graph.
- Use the cursor key [←] or [→] to move to any position.

→ The selection cursor position moves.

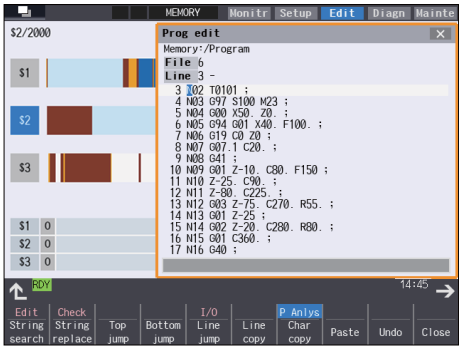
(3) Press the menu [Select].

→ The ONBT numbers are displayed.



Displaying the machining program

(1) Press the menu [Program display] on the ONB display screen to display the machining program. (*1)(*2)



(*1) The following table shows the relationship between program type at cursor position, the machining program and its display position on the program edit window.
For MTB macro/fixed cycle, a block before the display position might be displayed depending on the compression state of the graph display.
When MTB macro/fixed cycle are continued, the first display position is displayed. When MTB macro/fixed cycle are continued 50 times or more, the head of the main program is displayed.

Machining program type at cursor position	Displayed machining program	Display position
Main program	Main program	NB number position
Subprogram	Subprogram	NB number position
MTB macro	Source machining program	Previous block of source block
Fixed cycle	Source machining program	Previous block of source block

(*2) When the menu [Program display] is pressed, the machining program is displayed by referring to the ONB numbers in the machining program display. When the cursor position is changed and then the menu [Program display] is selected without pressing the menu [Select], the machining program may be displayed at a different position from the cursor position.

3.12 Defining Variables

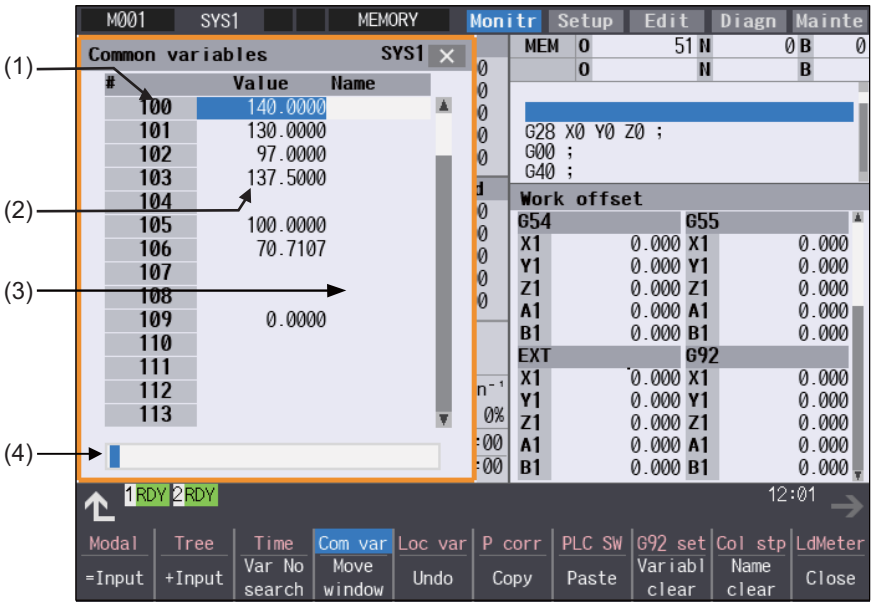
Refer to "Programming Manual" for more details on variables.

3.12.1 Common Variables

Monitr – #500 Com var

Select [Com var] on the monitor (Monitr) screen to set or display the details of the common variables.
If there is a common variable command (*1) 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.

(*1) The variable names can be set for common variables #500 to #599.



Display items

<table>
<tr><th colspan="2">Display items</th><th>Details</th></tr>
<tr><td>(1)</td><td>Variable number</td><td>This is the common variable No.
There are common variable 1(common to part systems) and common variable 2 (for each part system) (Refer to the table below).
The #100 variables appear at the head when the power is turned ON.
If "#1052 MemVal" is set to "1" (Designate No. of common variable common to part systems), "*" is attached to the variable that is common to part systems. (Note that if there is only one part system, "*" will not appear.)
<table>
<tr><th>Part system</th><th>Number of sets</th><th>Common variable 1 (common to part systems)</th><th>Common variable 2 (for each part system)</th></tr>
<tr><td rowspan="4">One part system</td><td>200 sets</td><td>500 to 599</td><td>100 to 199</td></tr>
<tr><td>600 sets</td><td>500 to 999
100100 to 800199 (*2)</td><td>100 to 199</td></tr>
<tr><td>700 sets</td><td>400 to 999 (*1)
100100 to 800199 (*2)</td><td>100 to 199</td></tr>
<tr><td>8000 sets</td><td>400 to 999 (*1)
100100 to 800199 (*2)
900000 to 907399 (*3)</td><td>100 to 199</td></tr>
<tr><td rowspan="2">Multi-part system (n = number of part systems)</td><td>600+100×n set(s)</td><td>400 to 999 (*1)
100100 to 800199 (*2)</td><td>100 to 199 ×n</td></tr>
<tr><td>7900+100×n set(s)</td><td>400 to 999 (*1)
100100 to 800199 (*2)
900000 to 907399 (*3)</td><td>100 to 199 ×n</td></tr>
</table>
</td></tr>
<tr><td>(2)</td><td>Variable value</td><td>The setting range is -999,999,999 to 999,999,999 or blank.
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
The minimum setting unit that can be set is 1.0000E-099 (99 digits below decimal point).
<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".
♦Common variables designated by the parameters #12111 to #12114 are unable to be set.</td></tr>
<tr><td>(3)</td><td>Variable name</td><td>A variable name can be assigned for #500 to #599.
Two-byte characters such as Chinese character can be displayed, however, only symbols and alphanumeric characters can be input on the screen. The maximum number of one-byte characters is 10.
<Note>
♦Common variables designated by the parameters #12111 to #12114 are unable to be set.</td></tr>
<tr><td>(4)</td><td>Input section</td><td>This displays details of the key input.</td></tr>
</table>

(*1) Address #400s common variable can be used only when the sets of common variable is "700 sets or more" and the parameter "#1336 #400_Valtype" is "1".

(*2) When the parameter "#1316 CrossCom" is set to "1", the common variables #100100 to # 800199 can be used common to part systems. The variables shown in the table below can be used common to part systems.

Common variables 1 (When "#1316 CrossCom" = "1")	
#100100 to #100199	Equivalent to # 100 to #199 in 1st part system
#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	Equivalent to # 100 to #199 in 5th part system
#600100 to #600199	Equivalent to # 100 to #199 in 6th part system
#700100 to #700199	Equivalent to # 100 to #199 in 7th part system
#800100 to #800199	Equivalent to # 100 to #199 in 8th part system

(*3) When the parameter "#1052 MemVal" is set to "1", you cannot use the variables #900000 to #907399 that are available when the specification "8000 sets of variables" is enabled.

Menus

Menus	Details
=Input	This executes an absolute input. If "#8930 Disable=INPUT:var" is set to "1", this menu cannot be selected.
+Input	This executes an incremental input.
Var No search	This designates the variable No. to be displayed at the head. Common variable 1 (common to part systems) #400 to #999, common variable 1 (common to part systems) #100100 to #800199, and common variable 2 (for each part system) #100 to #199 are displayed in each area. When another area is displayed, press this menu and specify the common variable No.
Move window	This moves the display position of the common variable window.
Undo	This returns the last rewritten data to its previous 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. <Note> ♦When the parameter "#8939 Undo confirm msg" is set to "1", the menu is highlighted while the operation message is displayed.
Copy	This copies the variable value and variable name at the cursor position.
Paste	This pastes the copied variable value and variable name.
Variabl clear	This clears the variable value(s) at the cursor position or in the designated range. <Note> ♦The contents are not cleared to "0". The state with no data is entered. ♦Common variables designated by the parameters #12111 to #12114 are unable to be cleared.
Name clear	This clears the variable name(s) at the cursor position or in the designated range. <Note> ♦Common variables designated by the parameters #12111 to #12114 are unable to be cleared.
Close	This closes the pop-up window and quits this function.

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

(1) Press the menu [Var No search].

(2) Designate the variable No.
102 [INPUT]

The cursor can be moved to 102 using the [↑], [↓], PAGE, or PAGE key.

(3) Input a numeric value.
135 [INPUT]

The value can also be designated by pressing the menu [=Input] instead of the [INPUT] key.
The value can also be input as an exponential.
(1.35E2 [INPUT] or 1.35E2 [=Input].)

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) Input a numeric value.
1.234 [+Input]

➡ The input value added to the original value is displayed. (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.

(2) Press the →| key.

(3) Input the variable name.
COUNTER [INPUT]

Note

(1) The variable name can be set only for variable No. 100 to 199 or 500 to 599.

Operation method (Copying and pasting the common variable)

(1) Select the variable to be copied.

One of the following methods can be used to select the variable.

- Using the cursor movement key [↑], [↓], PAGE, or PAGE, move the cursor to the variable to be selected.
- Press [Var No search] and set the variable No. in the input area, and press the [INPUT] key.

(2) Press the menu [Copy].

(3) Select the variable to be pasted, and press the menu [Paste].
The copied variable is held until a variable is newly copied.

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

(1) Press the menu [Variabl clear].

(2) Input the No. (range) of the variable to be erased.
102/104 [INPUT]
To erase only one variable, input one variable No.
(102 [INPUT])

(3) Press the [Y] key.

Note

(1) The variable at the cursor position will become the target of the erasing if the [INPUT] key is pressed without designating a variable No.

(2) The variables in the range below are the erasing target when /E is designated.

Input example	Erasing range	Supplement
100/E	100 to 199 100100 to 100199 (when the displayed part system is 1)	Only common variable in the displayed part system is cleared.
400/E	400 to 999 100100 to 800199 900000 to 907399	Range to be cleared varies depending on the number of common variable sets.

(3) If you include a variable between 100s and 500s in the range designation, the erasing is not performed.
(Example) When you set "110/510", "setting error" is displayed and the variables are not erased.

(4) When the variables protected from setting operation by the parameters #12111 to #12114 are included, the erasing is performed excluding the protected variables. When the whole range designated by the variables clear is within the protected range, "setting error" is displayed.

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

(1) Press the menu [Name clear].

(2) Select the variable , and press the [INPUT] key.
509 [INPUT]

(3) Press the [Y] key. → The variable name for 509 only is erased.
The variable value is not erased.

Note

(1) The variable at the cursor position will become the target of the erasing if the [INPUT] key is pressed without designating a variable No.

(2) The variables in the range below are the erasing target when /E is designated.

Input example	Erasing range	Supplement
100/E	100 to 199	Only variable name in the displayed part system is cleared.
500/E	500 to 599	

(3) Variable names protected from setting operation by the parameters #12111 to #12114 are not erased.

Switching the variable No.

Methods to switch the display areas are as follows.

(1) Use the menu [Var No search].

(2) Use the page up/down keys.

- If you press the next page key during displaying the last page of the area, the first page of the next area will be displayed.
- If you press the previous page key during displaying the first page of the area, the last page of the previous area will be displayed.
  (Example) If you press the next page key during displaying the last page of the common variable 2 (for each part system) (#100 -), the first page of the common variable 1 (common to part systems) (#500 -) will be displayed.

(3) Use a cursor key

- When a cursor key is placed at the biggest variable No. in the area, the first page in the next area will be displayed by pressing [↓] or [→|].
- When a cursor key is placed at the first variable No. in the area, the biggest variable No. in the previous area will be displayed at the top by pressing [↑] or [|←].

(4) Use a scroll bar

- The first page of the next area will be displayed by pressing ▼, while the last page of the area is displayed.
- The biggest variable No. in the previous area is displayed at the head by pressing ▲, while the first page is displayed.

Variable numbers cannot be displayed continuously if another display area is inserted in between.

Switching the window display position

Press the menu [Move window] to change the currently displayed window position.

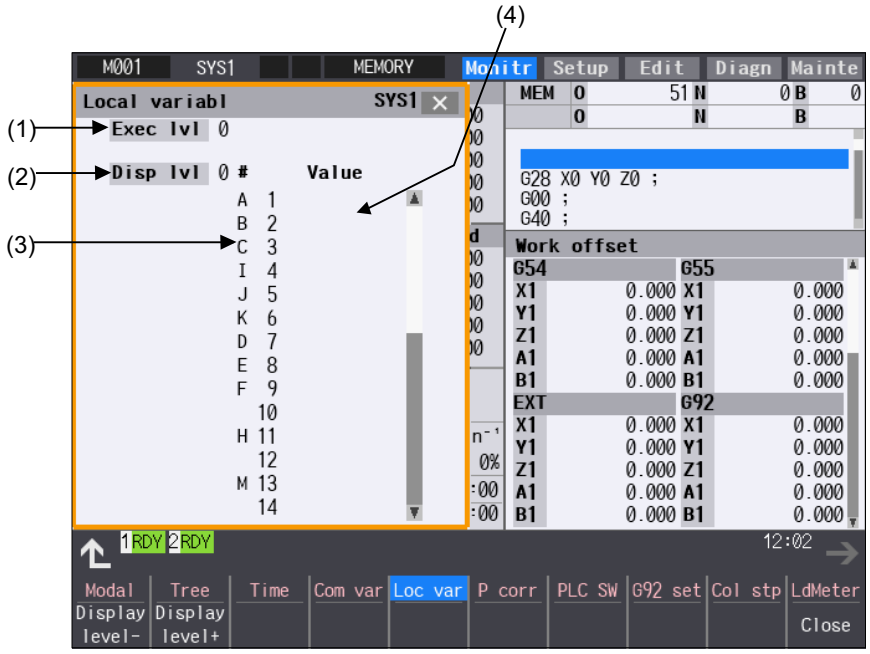
(1) Press the main menu [Com var]. → The common variable window is displayed in the pop-up window on the left of the screen.

(2) Press the menu [Move window]. → The common variable window will be move to the right.
The window position will be switched every time the [Move window] is pressed.

3.12.2 Local Variables

Monitr – Loc var

Select [Loc var] on the monitor (Monitr) screen, and 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.



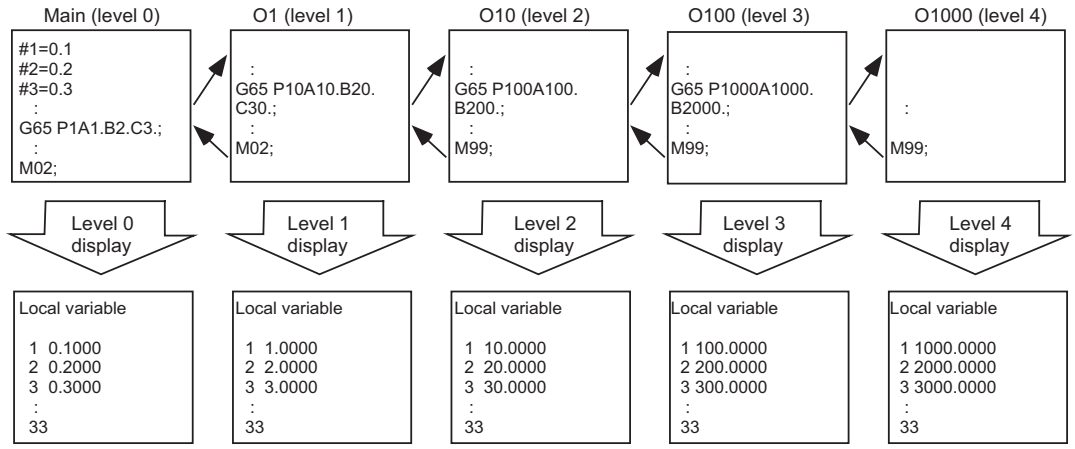
Display items

Display items		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 number	This displays the local variable No. The alphabetic character before the local variable No. is the argument code. G, N, O 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

Menus	Details
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.
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.
Close	This closes the pop-up window and quits this function.

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.

Operation method (Displaying the next level)

When the current display level is 0

(1) Press the menu [Display level +]. ➡ The level 1 local variables appear from the head.

When the menu [Display level +] 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)

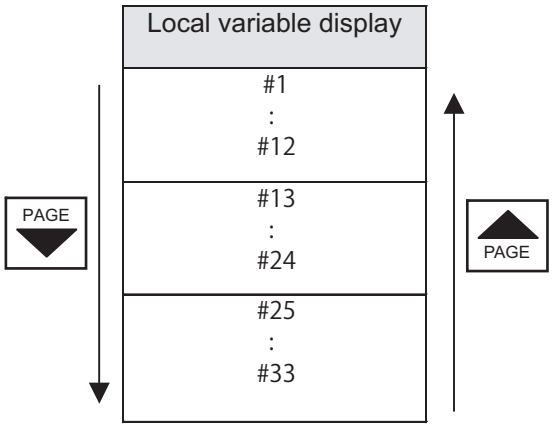
When the current display level is 2

(1) Press the menu [Display level -]. ➡ The level 1 local variables appear from the head.

When the menu [Display level -] 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 up/down keys (previous page PAGE, next page PAGE) are pressed.



4

Checking a Machining Program on Screen

This chapter explains checking the machining program. Generally, operations are performed on the edit (Edit) and monitor (Monitr) screen.

4.1 Graphic Check (2D)

Edit – 

Graphic check (2D) is a function that allows you to check the machining program movement path only by drawing without automatic operations. Therefore, the graphics can be drawn at a high speed and high accuracy, and the machining program can be checked.
Press the menu [Check] on the monitor or edit screen to switch to the graphic check screen.
The graphic check screen allows you to check the currently displayed part system program. In the 2-part system or more, press the part system switch key $\leftrightarrow$ to switch the display to the target part system before starting checking.
Note that program check is carried out in the 2-part system or more regardless of the command such as the timing synchronization between a hidden part system and another part system.
Using the menu [All display], normal display and whole display modes can be switched.

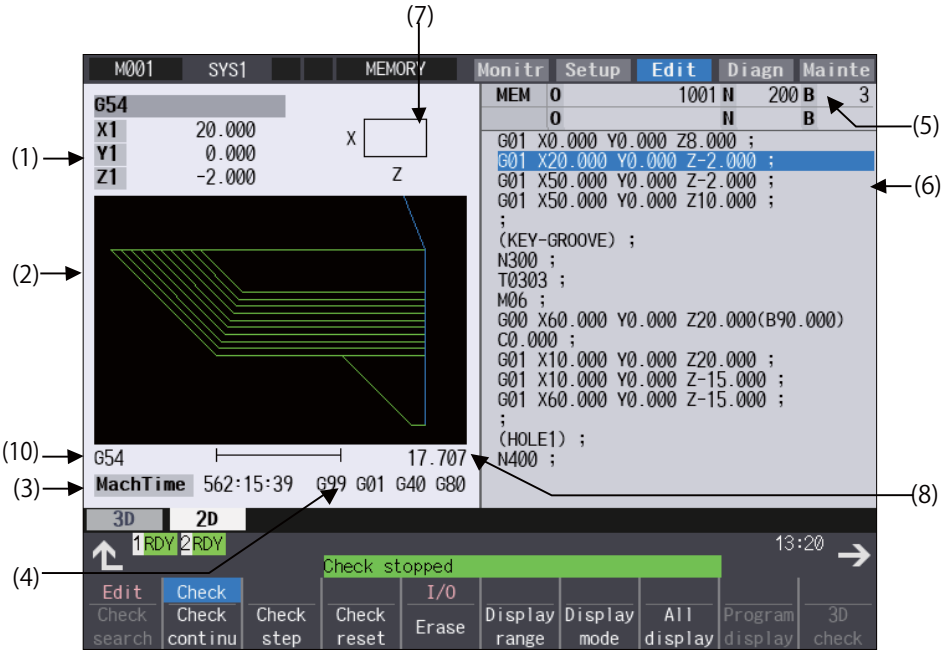
Note

(1) When changing the screen to the edit screen, display depends on the previous checking type (2D/3D). If 3D graphic data appears, press the menu [2D check].
However, if the previous two-dimensional check screen is displayed during tracing, the display switches to the 2D check screen only when tracing is turned OFF after a message was displayed to check that tracing is turned OFF.

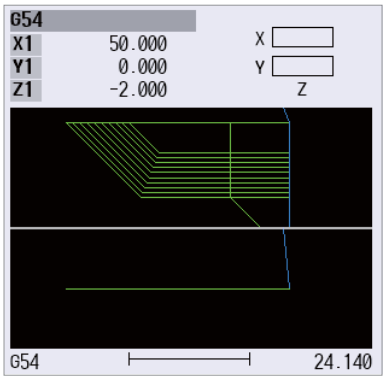
[Terms]

The following descriptions are the meanings of the terms used in this section.
Foreground: Active state operated by the user such as automatic operation on the NC
Background: Inactive state operated in the back of foreground

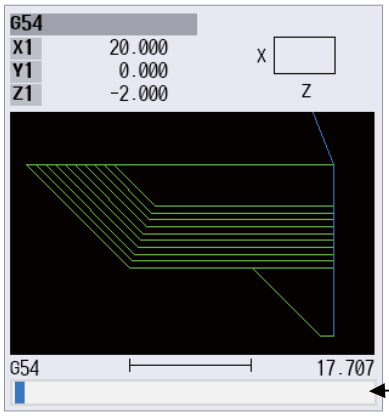
■Normal display ([All display] is OFF)



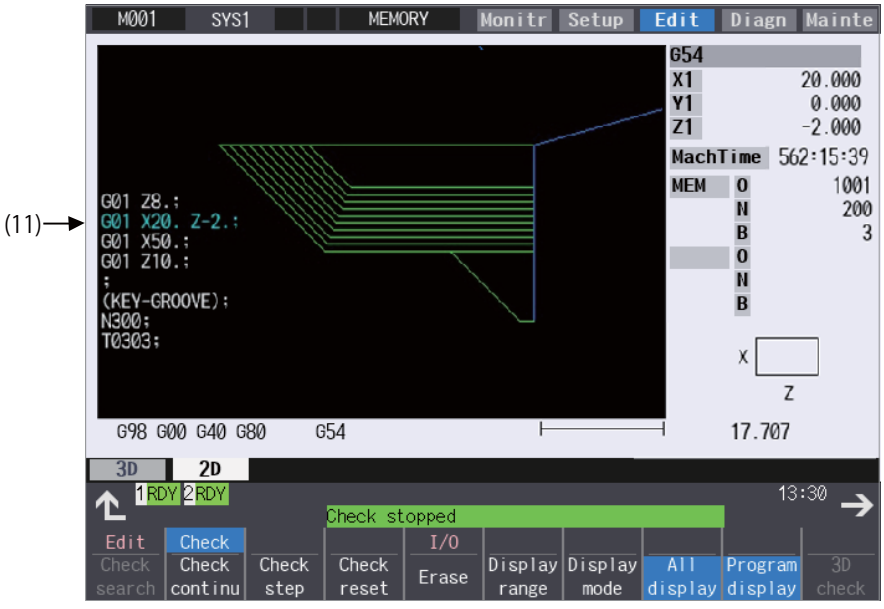
<Drawing area: 2-plane>



<Input area displayed>



■Full-screen display ([All display] is ON)



Display items

Display items		Details
(1)	Check counter	This displays the axis counter for the check drawing. When the drawn coordinate system is set to all workpiece coordinates with the menu [Drawing System], the counter name and value are switched according to the setting values of the parameters "#1231 set03/bit1" (Switch graphic coordinates) and "#1231 set03/bit5" (Switch graphic check counter display) as follows: #1231 set03/bit1=0 and #1231 set03/bit5=0: Machine position #1231 set03/bit1=0 and #1231 set03/bit5=1: Workpiece coordinate position #1231 set03/bit1=1 and #1231 set03/bit5=0: Tool position (machine coordinate system) #1231 set03/bit1=1 and #1231 set03/bit5=1: Tool position (workpiece coordinate system) When the selected drawn coordinate system is other than all workpiece coordinates, the displayed counter is the workpiece coordinate position or the tool position (workpiece coordinate system) regardless of the parameter "#1231 set03/bit5". The part system common axis name ("#1022 axname2") which conforms to the basic axis name set in the parameters "#1026 base_I", "#1027 base_J", and "#1028 base_K" are displayed for the three axes to display. <Note> ♦The settings of the parameters "#1026 base_I", "#1027 base_J" and "#1028 base_K" cannot be changed during the execution of 2D program check. The message "Program checking" appears.
(2)	2D drawing area	This is the area where the tool path is drawn. The drawing viewpoint can be moved, and the drawing size can be enlarged and reduced. The display for the path to be drawn will be switched among "only tool center path/ only program path/both tool center path and program path" by setting parameters "#1231 set03/bit1" (Switch graphic coordinates) and "#1231 set03/bit2" (Switch graphic check trace). #1231 set03/bit1=0 and #1231 set03/bit2=0: Draws both paths #1231 set03/bit1=0 and #1231 set03/bit2=1: Draws only tool center path (machine position) #1231 set03/bit1=1 and #1231 set03/bit2=0: Draws both paths #1231 set03/bit1=1 and #1231 set03/bit2=1: Draws only program path (tool position) <Note> ♦If the radius compensation has not been set, only tool center path (machine position) is drawn even when the both paths are set to be drawn. The zero point mark display position is switched between the machine coordinate zero point and workpiece coordinate zero point by the parameter "#1231 set03/bit4" (Switch zero point mark display position).
(3)	Machining time display	The time required for machining is calculated and displayed.
(4)	G modal display for check	<M system> The following items are displayed from the left. ♦Absolute/incremental (G90, G91: Group 3) ♦Operation mode (G00, G01, etc.: Group 1) ♦Tool compensation (G40, G41, G42: Group 7) ♦Fixed cycle (G70 etc.: Group 9) The item to be displayed is changed depending on the status of grpup 24 command modal, when the G code dynamic switching specification for M/L system is valid. <L system> The following items are displayed from the left. ♦Feed per minute/Feed per revolution (G98, G99: Group 5) ♦Operation mode (G00, G01, etc.: Group 1) ♦Nose R compensation (G40, G41, G42: Group 7) ♦Fixed cycle (G70 etc.: Group 9) The items to be displayed vary depending on the status of group 24 command modal.

Display items			Details
(5)	Currently executed machining program	Main O10...	This displays the device name, program No., sequence No., and block No. currently being executed. When they exceed the displayable number of characters (12 characters), the excess is abbreviated and displayed (*).
		Sub O1234...	This displays the sub-program device name, program number, sequence number, and block number for the sub-program currently being executed. When they exceed the displayable number of characters (12 characters), the excess is abbreviated and displayed (*).
(6)	Buffer display		This displays the contents of the machining program currently being executed. The block being executed is highlighted. When the parameter "#19431 Num display in prg" is set to "1", the variable, operation or function in the numerical part that follows each address is replaced with the calculated value and displayed for the current block and the next block.
(7)	Display mode		This shows the plane to be displayed for the 3-dimensional display. It can be selected with the menu [Display mode].
(8)	Scale		This shows the display range scale. The unit is set to [mm/100pixcel] for metric system and [inch/100pixcel] for inch system. Use the menu [Zoom in] or [Zoom out] to change the scale value.
(9)	Input section		This displays details of the key input when setting the scale value and the display axis name, and when setting the extended workpiece coordinate system as the coordinate to draw. Select the menu [Display range] and [Display mode] to display the input area. With the [INPUT] key, the input area is hidden for the scale value and display axis name settings. With the menu [Drawing system], the input area is displayed in the disabled state. With the menu [Coord G54.1P], the input area is enabled and the input area is hidden using the [INPUT] key.
(10)	Coordinate system Drawing coordinate settings		This displays the currently selected drawing coordinate settings.
(11)	Program display		The contents of the machining program that is in the process of drawing are displayed. The block currently being executed is displayed in blue characters. For 8.4-type/10.4-type display unit, the program display is not displayed on the normal display mode.

Menus

Menus	Details
Check search	This performs a check search. Select a program to be checked from the program file list displayed in the pop-up window.
Check continu	This checks the program continuously.
Check step	This checks the program per block.
Check reset	This resets 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. When the graphic is drawn after setting "#1231 set03/bit2" (Switch graphic check trace) to "1", pressing this menu again erases whichever a path of rapid traverse or cutting feed at once.
Display range	This changes the graphic drawing display range. Press this menu to display the input area for setting the scale value. When the display range is changed, the graphic data displayed on the screen is erased.
Display mode	This changes the drawing plane. Press this menu to display the input area for setting three axes of drawing targets. There are three types of graphic drawing mode: 1-plane, 2-plane, and 3-dimension. This changes the drawing plane and erases the graphic data displayed on the screen.
All display	This switches the normal display mode and the full-screen display mode.
Program display	This displays the machining program being checked on the graphic drawing area. For 8.4-type/10.4-type display unit, this menu can be selected when the menu [All display] is active.
3D check	This changes the program check (3D) display. This menu cannot be selected while the check is performed or interrupted.
Opt Skip	Select whether to execute a block that has a slash "/" in the program to be checked. Selecting "ON" skips a block with a slash "/" to carry out drawing. Selecting "OFF" also draws a block with a slash "/".
Rotate	This sets the viewpoint angle for the 3-dimension display mode. When the viewpoint angle is changed, the graphic data displayed on the screen is erased. However, this menu cannot be used for other than the 3-dimension display mode.
Regist. Image	This associates a graphic check image with a machining program and save the program.
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.
Drawing System	This sets the name and values of the counter to display in the graphic check, and the coordinate system to be drawn.
M0/M1 neglect	Designate whether to stop program check when M0/M1 is commanded. If [M0/M1 neglect] is ON, program check does not stop even when M0/M1 is commanded. If [M0/M1 neglect] is OFF, program check stops when M0/M1 is commanded.
Auto Graphic	After the scale is automatically adjusted so that the entire machining program can be displayed in the 90% of the drawing area, the graphic is drawn on the center of the drawing area. It is automatically adjusted in the following cases for 1-plane display mode. •When the program is selected •When display size is switched (normal display <-> full display) •When the part system is changed •When the display range is changed When auto graphic is valid, the menu is highlighted.

Menus	Details
Grid display	Whether to display or not the grid line in the drawing area for the 1-plane display mode can be selected. When the grid view is enabled, the scale value and scale bar size are obtained as the dimension value per grid. Enabling the grid view displays the scale extension line. Disabling the grid view erases the scale extension line. When the grid display is valid, the menu is highlighted.
Next axis	This changes the axis to be displayed on the counter. This switches between "Basic axes IJK" and "C axis". <Note> ♦This menu cannot be selected in the following cases: - When the control axis address other than C is specified in the parameter "#19405 Rotary ax drawing". - When C axis is specified in the user parameter "#1026 base_I" (Base axis I), "#1027 base_J" (Base axis J), or "#1028 base_K" (Base axis K). - When the "Graphic check/trace rotary axis drawing" specification is invalid.
Re-do scale	After the scale of auto graphic is automatically re-adjusted, the graphic is drawn on the center of the drawing area. This menu is valid when auto graphic is valid. This menu is used when the graphic cannot fit the scale of auto graphic by changing the data which affects scale such as tool compensation, workpiece coordinate system offset or sub-program.

Note

(1) The menus [Check search], [Check continu], [Check step] and [Check reset] cannot be selected while the check is performed or interrupted.
Menu can be selected after [Check reset] is pressed.

(2) The menus [Check search], [Check continu], [Check step], and [Check reset] are grayed out during automatic operation in trace mode, and the 3D solid check is disabled. If the program is not in the process of automatic operation, a check is possible only for 3D solid.

(3) The menus [M0/M1 neglect] and [Regist. Image] are grayed out during continuous check, and the menus cannot be selected.

(4) The menu [Re-do scale] is grayed out when the auto graphic is invalid, and the menu cannot be selected.

Sub-menus of [Check search]

Menus	Details
Memory	This selects the device for searching for the program. When a device with directory is selected, the root is selected first.
Memory2	
HD	
Memory card	
DS	
USB Memory	
Top jump	Display starts from the first page of the directory/list of files. The cursor moves to the top line.
Bottom jump	The last page of the directory/list of files is displayed. The cursor moves to the bottom line.

Menus	Details
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.
Preview	This switches to show or hide the preview (detailed information).
Change Disp	This switches between list display mode and thumbnail display mode.
List update	This updates the list contents. (The latest contents of the currently selected device and directory are listed.)
Sort change	This changes the method of sorting the list.
Close	This closes the pop-up window and quits this function.

Drawing color during check

The path of the tool center path (machine position) is drawn during the check. However, if a compensation is applied on the program being checked, the program path (tool position) and tool center path (machine position) are drawn.
While the compound type fixed cycle for turning machining I (G70, G71, G72, or G73) is commanded in L system, the program path (tool position), tool center path (machine position) and the path before tool shape compensation for turning machining are drawn.

The colors of the lines displayed during the check are expressed as follows.

		With compensation	Without compensation
Manual feed		-	-
Rapid traverse	Program path	Solid line (white)	-
	Tool center path	Solid line (blue)	Solid line (blue)
	Path before tool shape compensation for turning machining (Compound type fixed cycle for turning machining I)	Solid line (red)	Solid line (red)
Cutting feed	Program path	Solid line (white)	-
	Tool center path	Solid line (green)	Solid line (green)
	Path before tool shape compensation for turning machining (Compound type fixed cycle for turning machining I)	Solid line (red)	Solid line (red)

4.1.1 Checking Continuously

There are three types of method for checking the program.

- Checking the specified program
- Checking the program currently opened on the edit screen (Edit)
- Checking the program searched on the monitor screen (Monitr)

Operation method (Selecting a program to be checked)

Checking the specified program

(1) Press the [$↔$] key to switch the part system.
In addition to the above, the PLC signal and touch-screen operation are provided as the part system switching method.

(2) Press the main menu [Check] on the edit screen.
Press the menu [2D Check] when the program check (3D) appears.

(3) Press the menu [Check search].
If you press the menu [Comment nondisp] in the second page, the comment is hidden.

(4) Select the device.
(Example) Menu [Memory]

→ The selected device name and directory (memory: /program) appear in the device name and directory display columns.
For the devices other than NC memory or NC memory 2, root directory is selected.

(5) (For the devices other than NC memory or NC memory 2)
Use the [↑], [↓], [PAGE↑], or [PAGE↓] key to align the cursor with a directory of the program.

Press the [INPUT] key to move to inside the directory.

<Note>

- If the list contents differ from the actual device or directory, press the menu [List Update].
- Press the menu [Sort change] to change the sort. Each time the menu key is pressed, the sort method shifts in the following order: file names in ascending order -> file names in descending order -> dates and comments in ascending order -> dates and comments in descending order -> file names in ascending order.

(6) Press the [↑], [↓], PAGE or PAGE key to move the cursor to the program to be searched.
The name of the machining program to be edited can be input in the input area.
When the touchscreen display unit is used, you can also select a program by touch operation.

(7) Press the [INPUT] key.
When setting data exists in the input area, a search is performed in this data. This procedure is also used when performing an NB search.
(Example)
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

→ The check search is started.
The operation message "Search complete" 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.

Note

(1) The operation message "Sorting of tool data is in progress" is displayed when tool data is being sorted, and the check search is disabled.

(2) When you perform an operation search by designating a block No., it can be set by the parameter "#1760 cfgPR10/bit5" whether the block which consists of comment only is counted as one block. When the parameter is set to "0", the comment block is counted as one block.
Set the parameter to "1" to count the comment block as same as the automatic operation.

(Example) When an operation search is performed by the sequence No. 1 and the block No. 2

Sequence Nos. and the block Nos. vary as indicated in the following table depending on the setting of the parameter. Thus, the different block is searched.

When "#1760 cfgPR10/bit5" is set to "0", the block (a) is searched.

When "#1760 cfgPR10/bit5" is set to "1", the block (b) is searched.

Program	#1760 cfgPR10/bit5 = 0		#1760 cfgPR10/bit5 = 1	
	Sequence No.	Block No.	Sequence No.	Block No.
N01;	N1	B0	N1	B0
(COMMENT1);	N1	B1	-	-
G28 X0Y0Z0; .....(a)	N1	B2	N1	B1
G0 X10.; .....(b)	N1	B3	N1	B2
M30;	N1	B4	N1	B3

Checking the program currently opened on the edit screen

(1) Press the [$↔$] key to switch the part system.

In addition to the above, the PLC signal and touchscreen operation are provided as the part system switching method.

(2) Press the menu [Program check] on the edit screen.

Press the menu [2D Check] when the program check (3D) appears.

Checking the program searched on the monitor screen

(1) Press the [$↔$] key to switch the part system.

In addition to the above, the PLC signal and touchscreen operation are provided as the part system switching method.

(2) Press the menu [Check] on the monitor screen.

Press the menu [2D Check] when the program check (3D) appears.

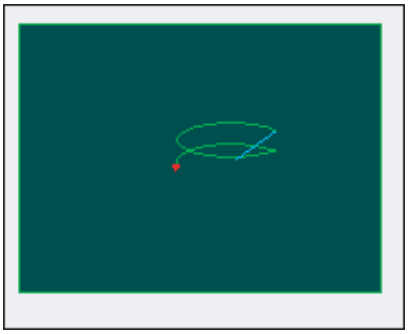
Operation method (Executing the program check)

(1) Press the menu [Check continu].

Behaviors during check operation

- Press the menu [Check step], the program check is carried out block by block starting from the next block.
- Press the menu [Check continu], the program check is stopped. Press the menu [Check continu] during block stoppage, the program check is carried out continuously.
- Press the menu [Check reset], the program check is reset after completion of the currently executing block.

→ 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 is not displayed.
The machining time is updated.
Program check is carried out for the currently displayed part system. As necessary, switch the part system to be displayed.



When the check is completed, a message appears and the mode returns to normal.
The menu highlight returns to normal.

When the setting of the bit1 or bit2 of parameter #1231 is changed, you can switch the drawing path. Refer to "4.1.12 Switching the Draw Path" for operation procedures.

Note

(1) The operation message "Sorting of tool data is in progress" is displayed when tool data is being sorted, and the check search is disabled.

4.1.2 Checking One Block at a Time

There are three types of method for checking the program.

- Checking the specified program
- Checking the program currently opened on the edit screen (Edit)
- Checking the program searched on the monitor screen (Monitr)

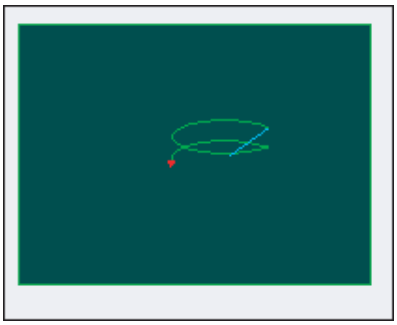
Perform the same procedure as "4.1.1 Checking Continuously" on selecting a program.

Operation method (Executing the program check)

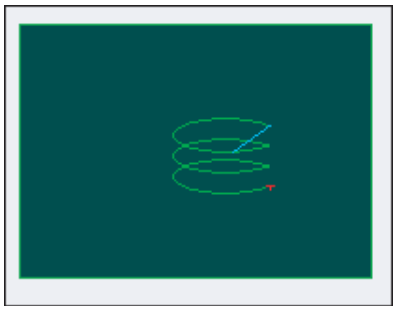
(1) Press the menu [Check step].

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>



(2) Press the menu [Check step].

The program check is executed for the single block.
When the check is completed, a message appears and the mode returns to normal.
The menu highlight returns to normal.

Behaviors during check operation

- Press the menu [Check continue], program check is carried out continuously up to the end of the program.
- Press the menu [Check reset], the program check is reset.

When the setting of the bit1 or bit2 of parameter #1231 is changed, you can switch the drawing path.
Refer to "4.1.12 Switching the Draw Path" for operation procedures.

Note

(1) The operation message "Sorting of tool data is in progress" is displayed when tool data is being sorted, and the check search is disabled.

4.1.3 Setting the Coordinate System to Draw

Press the menu [Drawing system] to select the coordinate system to draw from among the coordinate selection menus below. During the graphic check, it is possible to draw paths of the running programs of the selected coordinate system only. If the menu [All wk. coord] is selected, all paths of the program will be drawn. If any coordinate system menu other than [All wk. coord] is selected, paths of the program of only selected coordinate will be drawn. The default setting is [All wk. coord].

Note

(1) The coordinate system to draw that is set here is held even after turning the power ON again.

(2) When the parameter "#1231 set03/bit4" (Switch zero point mark display position) is set to "1", the zero point mark's position is not of the selected coordinate system to draw, but of the workpiece coordinate system or extended workpiece coordinate system specified by the machining program being checked.

(3) The coordinate system to draw is common between part systems.

(4) The drawing coordinate system settings of 2D and 3D graphic check are not linked.

(5) When any coordinate system menu other than [All wk. coord] is selected, if switching the coordinate system has not been commanded in the program to execute the program check, this draws the program performs drawing in the currently set coordinate system, that is not the correct position. Thus, make sure that the coordinate system is specified in the program.

Menus to select the coordinate system to draw

Menus	Description
All wk. Coord	This draws all workpiece coordinate systems. The display for the check counter switches by the setting of the parameter "#1231 set03/bit1" (Switch graphic coordinates) and "#1231 set03/bit5" (Switch graphic check counter display). In the area to show the setting of the coordinate system to draw, "All Work" is displayed. #1231 set03/bit1=0 and #1231 set03/bit5=0: Machine position #1231 set03/bit1=0 and #1231 set03/bit5=1: Workpiece coordinate position #1231 set03/bit1=1 and #1231 set03/bit5=0: Tool position (machine coordinate system) #1231 set03/bit1=1 and #1231 set03/bit5=1: Tool position (workpiece coordinate system)
G54 G55 G56 G57 G58 G59	This specifies the workpiece coordinate (between G54 and G59) as a coordinate system to draw. In the graphic check counter, the workpiece coordinate position or tool position (workpiece coordinate system) counter is displayed regardless of the setting of the parameter "#1231 set03/bit5" (Switch graphic check counter display). In the area to show the setting of the coordinate system to draw, the selected workpiece coordinate system is displayed.
Coord G54.1 P_	This specifies the extended workpiece coordinate (G54.1 Pn) as the coordinate system to draw. The extended workpiece coordinate (G54.1 Pn) can be selected for the type of coordinate system to draw by inputting the P No. The graphic check counter is changed to the workpiece coordinate position or tool position (workpiece coordinate system) counter regardless of the setting of the parameter "#1231 set03/bit5" (Switch graphic check counter display). In the area to show the setting of the coordinate system to draw, the selected workpiece coordinate system is displayed.

Select the type of workpiece coordinate system to draw (When selecting all workpiece coordinates, or coordinate system G54-59)

(1) Press the menu [All wk. coord]. → In the area to show the setting of the coordinate system to draw, "All Work" is displayed.

In the same manner, by pressing a menu between [G54] and [G59], in the area to show the setting of the coordinate system to draw, the selected workpiece coordinate system is displayed.

Select the type of workpiece coordinate system to draw (When selecting the coordinate system G54.1P)

(1) Press the menu [Coord G54.1P].

(2) Input "20", and press the [INPUT] key. → In the area to show the setting of the coordinate system to draw, "G54.1P20" is displayed.

Note

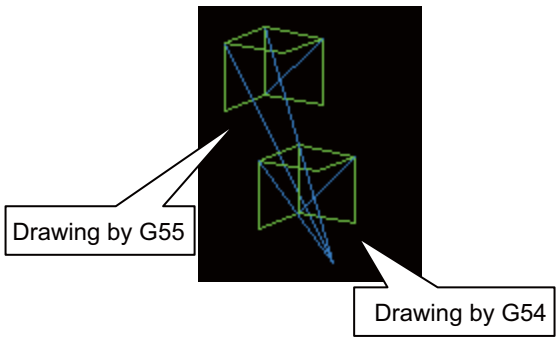
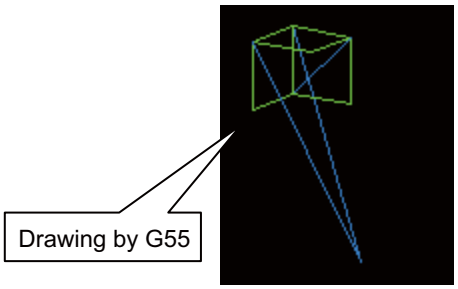
(1) When the [All wk. coord] or [G54] to [G59] is pressed while the menu [G54.1P] is being selected, the pressed menu is selected as the coordinate system to draw.

Difference of drawing depending on the selected workpiece coordinate system

The drawing of 2D check changes as follows depending on the selection of the sub menu of [Drawing system].

(Example)

[Workpiece coordinate]	[Machining program]
G54: X 100.000 Y 100.000 Z 0.000 G55: X 200.000 Y 200.000 Z 0.000	N1 G28 XYZ ; N10 G90 G54 G0 X100. Y100. Z100. ; N11 G91 G01 X-100. F500 ; N12 Y-100. ; N13 X100. ; N14 Y100. ; N15 Z-100. ; N16 Y-100. ; N17 Z100. ; N18 G0 Y100. Z-100. ; N19 G01 X-100. F500 ; N20 Z100. ; N30 G28 XYZ ; N40 G90 G55 G0 X200. Y200. Z200. ; N41 G91 G01 X-100. F500 ; N42 Y-100. ; N43 X100. ; N44 Y100. ; N45 Z-100. ; N46 Y-100. ; N47 Z100. ; N48 G0 Y100. Z-100. ; N49 G01 X-100. F500 ; N40 Z100. ; N50 G28 XYZ ; M30 ; % ;

Drawing coordinate system selection	Drawing at 2D check	
All wk. coord		Workpiece coordinates G54 and G55 both are drawn.
Workpiece coordinate system G55		Workpiece coordinate G55 is drawn. G54 is not drawn.

4.1.4 Drawing during a Program Check

During the program check, the tool center path (machine position) is drawn. However, if a radius compensation is applied on the program being checked, the program path (tool position) and tool center path (machine position) 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

4.1.5 Changing the Display Range

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

Operation method (Enlarging and reducing the drawing)

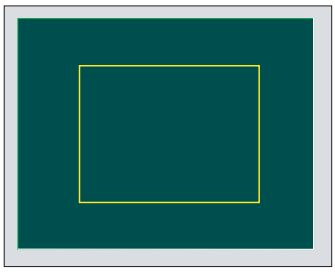
(1) Press the menu [Display range].

(2) Press the menu [Zoom In] or [Zoom out].

To enlarge the figure:
Press the menu [Zoom in] or [-] key.
To reduce the figure:
Press the menu [Zoom out] or [+] key.

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

→ The size of the yellow frame can be changed by using key operations.



(3) Press the [INPUT] key.

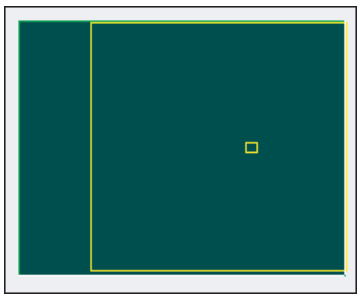
→ The display scale changes.
By changing the display scale, the graphic data displayed on the screen is erased.
When auto graphic is valid, the graphic data is automatically drawn again on the changed display scale.

Operation method (Changing the drawing display position)

(1) Press the menu [Display range].

(2) Press the menus [↑], [↓], [←], [→] 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.

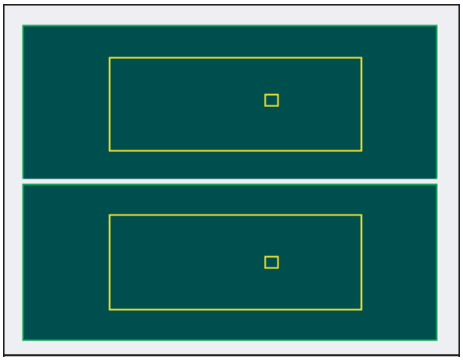


(3) Press the [INPUT] key.

➡ The display position changes so that the cursor position (□) is the center of the area.
When display position is changed, the graphic data displayed on the screen is erased.
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.

(2) The set display angle is maintained even after rebooting the machine.

(3) The graphics are not automatically drawn even when auto graphic is valid.

Operation method (Centering)

(1) Press the menu [Display range].

(2) Press the menu [Centering]. → The display position is changed so that the current machine position appears in the center of the drawing area. The graphic displayed up to that point is cleared and the drawing starts again from the current position (the center of the drawing area).
When auto graphic is valid, the graphic data is automatically drawn again in the changed display range.

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) When the menu [Centering] is pressed after the display scale has changed, only the centering is applied without changing the scale value.

Setting range of the scale value

#1003 Input setup unit	#1041 Initial inch	
	0 (mm)	1 (inch)
B	0.100 to 9999.999	0.0100 to 9999.9999
C	0.0100 to 9999.9999	0.00100 to 9999.99999
D	0.00100 to 9999.99999	0.000100 to 9999.999999
E	0.000100 to 9999.999999	0.0000100 to 9999.9999999

(3) The set scale value is maintained even after the power is turned OFF and ON.

(4) In the 2-plane display mode such as [XY/XZ], the change of the scale input range is applied for the upper and lower area. The operation method is the same as the 1-plane display mode.

4.1.6 Changing the Display Mode

There are three types of graphic drawing mode: 1-plane, 2-plane, and 3-dimension. Press the menu [Display mode] key, and by selecting one of the sub menus, the axis configuration for each plane changes, and the menu display returns to normal. Set the axis name in the input area and select the [INPUT] key to change three axes of the drawing targets.

Note

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

(2) These set display mode is maintained even after rebooting the machine.

(3) The display mode can be set independently for the trace function and program check function. These settings are not linked with one another.

(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.

(5) When you select the menu of the plane or cube which does not exist on actual axis configuration, the display mode is not switched.

(Example) In case that you select the [XY] or [XYZ] menu on the 2-axis (X and Z axes) configured machine

Display mode menu

Menus	Description
XY	This changes to the 1-plane display mode configured of X-Y. The X axis is displayed in the horizontal axis, and the Y axis is displayed in the vertical axis.
XZ	This changes to the 1-plane display mode configured of X-Z. The X axis is displayed in the horizontal axis, and the Z axis is displayed in the vertical axis.
YX	This changes to the 1-plane display mode configured of Y-X. The Y axis is displayed in the horizontal axis, and the X axis is displayed in the vertical axis.
YZ	This changes to the 1-plane display mode configured of Y-Z. The Y axis is displayed in the horizontal axis, and the Z axis is displayed in the vertical axis.
ZX	This changes to the 1-plane display mode configured of Z-X. The Z axis is displayed in the horizontal axis, and the X axis is displayed in the vertical axis.
ZY	This changes to the 1-plane display mode configured of Z-Y. The Z axis is displayed in the horizontal axis, and the Y axis is displayed in the vertical axis.
XY/XZ	This changes to 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.
YX/YZ	This changes to 2-plane display mode configured of Y-X and Y-Z. The Y axis is displayed in the horizontal axis, and the X and Z axes are displayed in the vertical axis.
XYZ	This changes to the 3D display mode. A cube is displayed on the lower right of the screen.

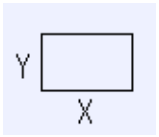
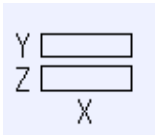
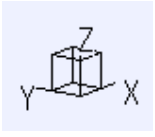
Changing the display axis name

(1) Set the axis name and press the [INPUT] key.
XYC [INPUT]

➡ XYZ appears the graphic display mode name. There are 3 drawing axes: X, Y, and C.
The X axis, Y axis and C axis are the three axes for drawing.

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)
		

4.1.7 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.

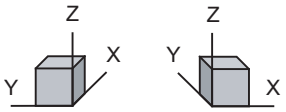
Note

(1) When the display angle is changed, the graphics displayed up to that point are erased.
(2) The set display angle is maintained even after rebooting the machine.
(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].

(2) Use the menus [↑], [↓], [←], [→] to adjust the display angle.
The cursor movement keys ([↑], [↓], [←], [→]) can also be used to rotate.
→ Rotate the cubic display in the lower right of the screen using the operation keys.



(3) Press the [INPUT] key.
→ The display angle changes.
By changing the display angle, the graphic data displayed on the screen is erased.

Note

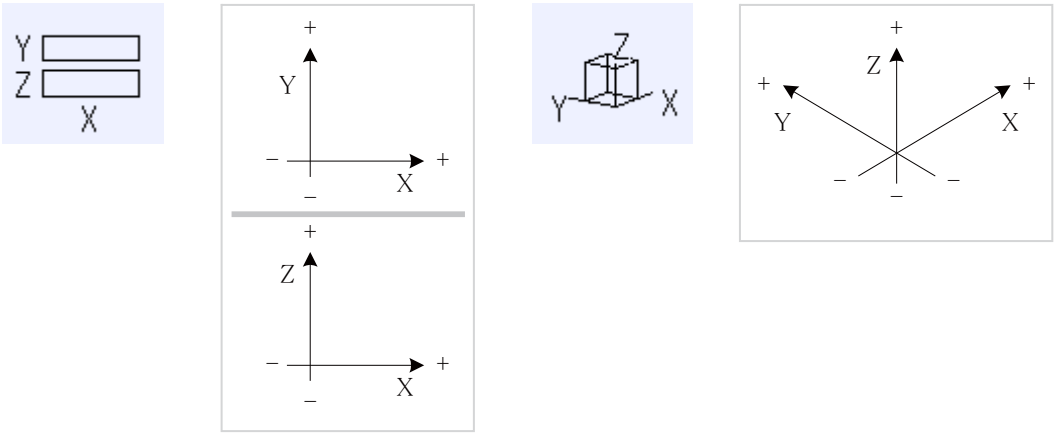
(1) The set display angle is maintained even after rebooting the machine.

4.1.8 Switching the Axis Movement Direction (+/-)

The path is drawn according to the axis movement direction designated by the parameter "#1495 grf_ax_direction". The movement direction can be set for each axis.

When "#1495 grf_ax_direction" is set to "0", the axis movement directions are as follows.

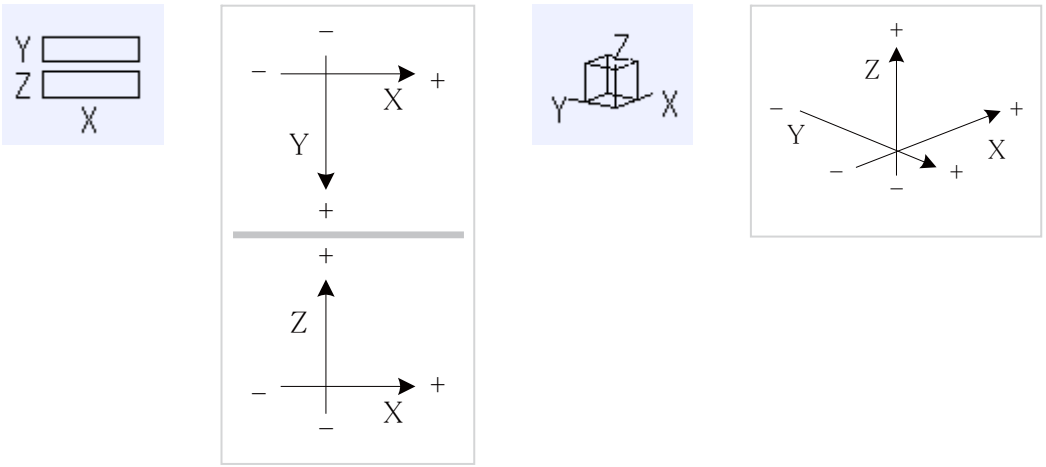
<2D mode> <3D mode>



When the axis movement direction is opposite direction, the axis direction movement of 2D or 3D mode are as follow.

(Example) When the Yaxis movement direction is set in the opposite direction

<2D mode> <3D mode>



4.1.9 Displaying the Machining Program on Checking

When pressing the menu [Program display], the contents of the machining program on checking is displayed on the drawing area.
When pressing the menu [Program display] again, the displayed program is erased.
For 8.4-type/10.4-type display unit, the menu is not displayed on the normal display mode.

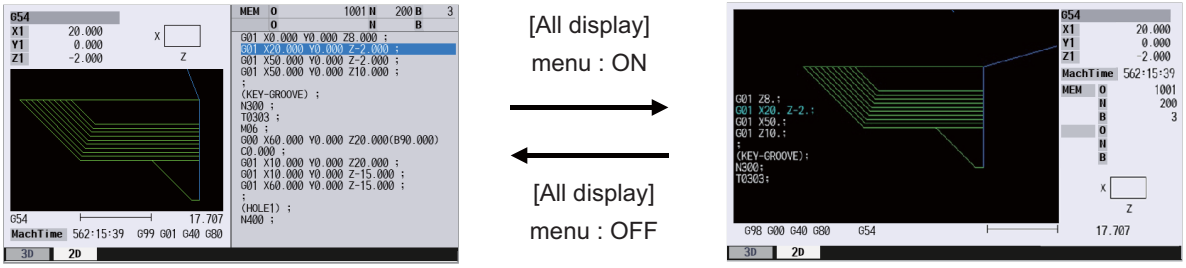
Note

(1) The selected display mode is maintained even after rebooting the machine.
(2) The program display mode is applied to the trace function and program check (2D) function commonly.
(3) For 8.4-type/10.4-type display unit, the menu [Program display] can be selected when the menu [All display] is active.
(4) The program display mode is applied commonly to all part systems.
(5) When the parameter "#19431 Num display in prg" is set to "1", the variable, operation or function in the numerical part that follows each address is replaced with the calculated value and displayed for the current block and the next block.

4.1.10 Switching to Full-screen Display Mode

Press the menu [All display] 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

Full-screen display mode

Note

(1) The selected display mode is maintained even after rebooting the machine.
(2) The full-screen display mode is applied to the trace function and program check (2D/3D solid) function commonly.
(3) The full-screen display mode is applied commonly to all part systems.
(4) When auto graphic is valid, the graphic is drawn after the drawing's scale is automatically adjusted to the scale to draw the entire machining program (the area of about 90% of the drawing area).

4.1.11 Setting Auto Graphic

If [Auto Graphic] is valid, a graphic is automatically drawn when a file is selected to check a graphic, the display size is switched (normal display <-> whole display), the part system is changed or the display range is changed.
When a file is selected, the part system is changed, or the display size is switched, a graphic is drawn in the center of the drawing area after the scale is automatically adjusted so that the entire machining program can be drawn in the 90% of the drawing area.

A drawing path is switched according to the setting of parameter "#1231 set03/bit1, bit2".
Refer to "4.1.12 Switching the Draw Path" for operation procedures.

(1) Press the menu [Auto Graphic].

(2) Press the menu [Check search]. → The check search list is displayed.

(3) Use the [↑], [↓], PAGE, or PAGE key to point the cursor with the target machining program.
(Or enter the target file name in the input area.)

(4) Press the [INPUT] key. → The check search is started.
The message "Search completed" appears when the search is completed.

When the check search is completed, the graphic previously displayed is erased and the auto-adjustment of the scale starts.
During the auto-adjustment, a message "Executing auto scale" is highlighted.

When the auto-adjustment of the scale is completed, "Checking" message is displayed and the graphic drawing is performed automatically.

When the graphic drawing is completed, the message "Program check completed" is displayed and the graphic drawing mode is canceled. The highlight of each menu returns to normal and the menus become selectable.

Precautions

(1) Auto graphic is executed only for the 1-plane display mode. The menu [Auto Graphic] cannot be selected for the other modes. Auto graphic mode always remains as it is in the 1-plane display mode.

(2) Once the auto graphic mode is set, it is retained after the power is turned ON again.

(3) The auto graphic cannot be set for the trace function.

(4) The auto graphic mode is common between part systems.

(5) Even the M00 or M01 command is issued, the program does not stop.

(6) If a program which contains M99 command is selected, auto adjustment of the scale will be executed repeatedly and the adjustment will not be completed with the "Executing auto scale" message blinking.
Do not select a program which contains M99.
Execute check reset when it happens.

(7) When a large capacity program is selected, it may take time to perform the auto graphic.

(8) If the display size is switched (between normal and full screen) or the display range is changed during program check, the auto graphic drawing is not carried out even if the auto graphic is valid. Auto graphic drawing is performed only when the figure is enlarged or reduced.

(9) If a check search is performed on a file which went through an auto-adjustment of the scale, only graphic drawing is performed as long as there is no changes in "a path and a file name (including a device)", "the date of updating", or "display mode (XY, YZ, XZ, etc.)". Maximum 12 files can be memorized, to which the automatic scale adjustment has been executed.
Therefore, if the 13th file is check-searched and the automatic scale adjustment is executed, the information of the first file is lost. When the first file is check-searched again, the automatic scale adjustment starts after check search has been completed.

(10) The automatic scale adjustment is interrupted if one of the following operations is taken:

- Press the menu [Check reset].
- Press the menu [Auto Graphic].
- The screen is changed to other than [2D check].
- A program error occurs.

(11) When the data related to the machining program (tool compensation, workpiece coordinate system offset, sub-program, etc.) drawn with the auto graphic once is changed, the shape of auto graphic may not fit on the screen. In this case, press the menu [Re-do scale] to re-adjust the scale.
The following situations are examples which the auto graphic shape does not fit on the screen.

- When the parameter "#1051 MemTol" (Tool compensation memory common for part systems) was set to "1", the tool compensation amount was changed and the part system was switched.
- After the tool compensation amount was changed, the operation search was performed on other machining program.
- The values were changed on the workpiece coordinate offset screen or common variable screen.
- The sub-program was edited.

4.1.12 Switching the Draw Path

The path to be drawn can be switched by designating a parameter.

■Parameter "#1231 set03/bit2" (Switch graphic check trace)

The number of the paths to be drawn can be switched. It is immediately activated after the setting.

Parameter value	Description
0	Both the tool center path (machine position) and program path (tool position) are drawn. However, the radius compensation is not reflected for the program path.
1	Only path designated with "#1231 set03/bit1" (Switch graphic coordinates) is drawn.

■Parameter "#1231 set03/bit1" (Switch graphic coordinates)

When the parameter "#1231 set03/bit2" (Switch graphic check trace)" is set to "1", either of the following drawn path can be selected. Selected drawn path is immediately activated after the setting is changed.

Parameter value	Description
0	Only the tool center path (machine position) is drawn.
1	Only program path (tool position) is drawn. The path does not reflect the radius compensation.

The counter display is switched depending on the combination of the setting values of the parameters "#1231 set03/bit1" and "#1231 set03/bit5". When the drawing coordinate system by the menu [Drawing system] is set for other than "All wk. coord", the counter of the workpiece coordinate position or the tool position (workpiece coordinate system) is displayed regardless of a parameter "#1231 set03/bit5".

#1231		set03/bit5	
		0	1
set03/bit1	0	Machine position	Workpiece coordinate position (*1)
	1	Tool position (machine coordinate system)	Tool position (workpiece coordinate system) (*1)

(*1) The name of the counter display is changed according to the currently executed workpiece coordinate system.

4.1.13 Drawing a Path Corresponding to the Rotary Axis (L System)

When the rotary axis (C axis) is designated with the parameter "#19405 Rotary ax drawing" in program check (2D), the path is drawn depending on the rotary axis. The rotary axis can be set for each part system.

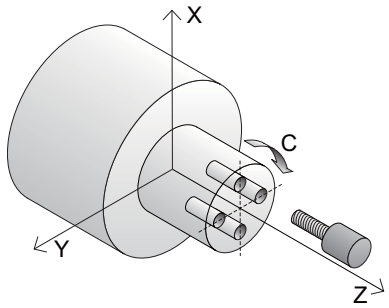
Note

(1) The "Graphic check/trace rotary axis drawing" option is also required to use this function.

(2) When the control axis address other than C axis is designated with the user parameter "#19405 Rotary ax drawing", the path corresponding to the rotary axis is not drawn.

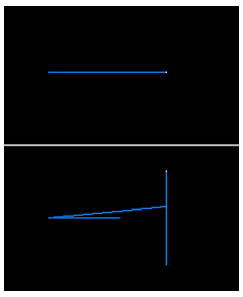
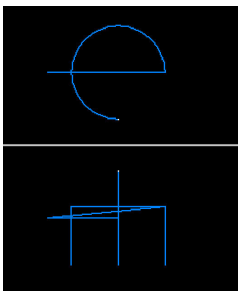
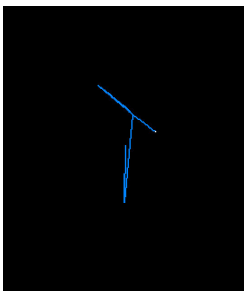
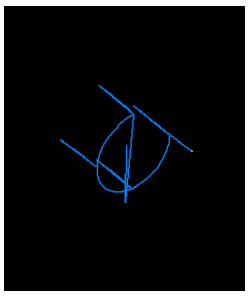
(3) The workpiece coordinate system must be set so that the center of the rotary axis becomes the coordinate system zero point.

Image of the actual tool and workpiece



Example of machining program

```
%
O0512
T1
G97S1000M74
M19
G28H0
G0C0
G0X50.Z5.
G98
G83Z-20.R-5.Q5000F300
G0C90.
G83Z-20.R-5.Q5000F300
G0C180.
G83Z-20.R-5.Q5000F300
G0C270.
G83Z-20.R-5.Q5000F300
G80
G0Z20.
G28U0.M75
M18
M30
%
```

Drawing image for 2D mode		Drawing image for 3D mode	
When the rotary axis drawing is invalid	When the rotary axis drawing is valid	When the rotary axis drawing is invalid	When the rotary axis drawing is valid
			

4.1.14 Associating Graphic Check Images with a Machining Program

The graphic check image displayed in the drawing area can be saved as associated with a machining program.

Operation method (Registering image data)

(1) Press [Regist. Image].

➡ The message "Registration completed" will appear when image data registration is complete.
If existing image data is already registered, the confirmation message "Overwrite this file?(Y/N)" will appear.

For information on how to erase the associated image data, see the entry for "Operation method (Erasing image data)" in "3.5 Erasing a File".

Note

(1) If saving to an external device (other than NC memory or NC memory 2), the image data will be stored in the "IMG" directory under the directory containing the machining program. If the directory does not exist, the system will automatically create one and save the image data to it.

(Example) If the machining program is found in the "DS:/FILE" directory, image data will be stored in the "DS:/FILE/IMG" directory.

(2) Each device will have limited space available for registering images. If the device capacity is exceeded, the operation message "Memory over" will appear and no additional image data can be registered.

Operation method (Displaying attached image data)

(1) Press [Check search].

(2) Select the device.
(Example) [Memory]

➡ The selected device name and directory (memory:/program) appear in the device name and directory display columns.
For devices other than NC memory and NC memory 2, the root directory is selected.

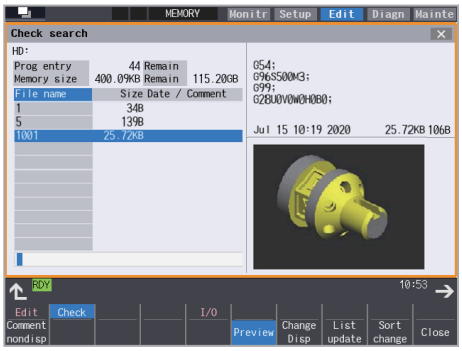
(3) (For devices other than NC memory or NC memory 2)
Press the [↑], [↓], PAGE, or PAGE key to align the cursor with the target directory.

If the list contents differ from the actual device or directory, press [List update].

(4) (For devices other than NC memory or NC memory 2)
Press [INPUT].

(5) Press the [↑], [↓], PAGE or PAGE key to align the cursor with the machining program to which the image data was registered.

(6) Press [Preview]. → The image data is displayed.



4.1.15 Setting the Drawing Speed of Program Check

Check speed of graphic check can be accelerated. When accelerating, set the parameter "#8997 Hi-speed grph chck" to "1" to "3" (Level 1 to Level 3). However, the shape of workpiece drawing becomes the one taking inner route than the path commanded by the program.

When you want to confirm the shape with high-accuracy, disable the high-speed graphic check.

High-speed graphic check invalid	High-speed graphic check valid
The shape can be confirmed with high-accuracy. The check speed is low. 	The check can be performed at high-speed. Minute shape collapses. 

Note

(1) The setting of high-speed graphic check is not reflected while [Check continu] or [Check step] is operated in the background.

(2) High-speed graphic check is valid only for check method I.

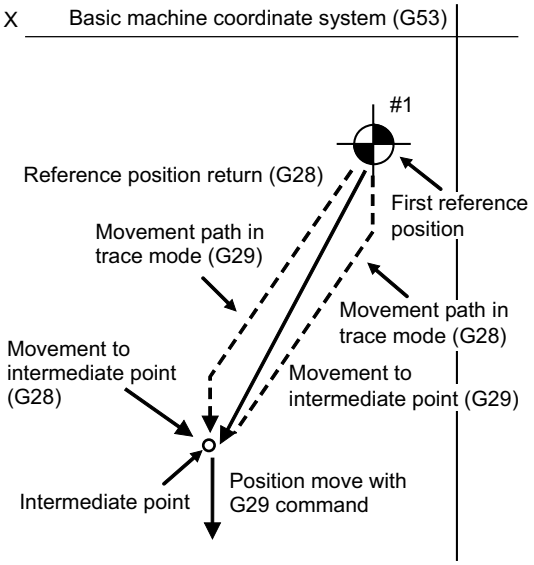
4.1.16 Availability of Check Mode with Other Functions

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	A mirror image is applied and a check drawing is made at the setting status prior to starting the check.
Mirror image by external input	The signal is reflected, and a check drawing is carried out (Foreground). The signal is ignored, and a check drawing is carried out (Background).
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)	If you press the cycle start button during checking, the operation alarm "M01 program check mode" is displayed (foreground) while the button is held down. The operation can be started automatically even during check drawing (background).
Auto-restart	The check is completed at M02/M30 and is not restarted.
Workpiece coordinate offset	Offset is enabled (the target is drawn at the position determined with the designated offset amount).
Manual mode/handle	If the mode is switched to the manual mode during checking, the operation alarm "M01 program check mode" is displayed (foreground). During checking, you can switch to the manual mode, HANDLE, to move the machine. However, the drawing position of the check is not affected by the mode change (background).
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. •Macro interface input/output •Single block stop •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 and 2nd miscellaneous function commands	M, S, T and 2nd miscellaneous function command signals commanded by the program during the check are not output. Neither do they display on the operation screen etc. The auxiliary function command by manual operation on the position display screen is output even during checking.
G command modal	G modal displays on the modal display screen while the check drawing is being made.
Optional block skip	The setting status when starting the check drawing are used (foreground). Use the menu to switch ON and OFF. The optional block skip state is held until the power is turned ON again, which is common to part systems. However, when the power is turned ON again, the mode is always turned OFF (background).
Spindle position control (spindle/C axis control)	When the program command method is selected, the spindle/C axis moves in the C axis mode.

Note

(1) 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.



4.1.17 Handling of Variable Commands, Programmable Parameter Input, and Programmable Compensation Data Input

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. The setting commands, which are executed to various data during program check, are reflected on only drawing of background check. The actual machining operation is not affected.

Meaning of symbols in the table
○: The value returns to the status prior to starting the program check (Actual machining data is not rewritten).
×: The value does not return to the status prior to starting the program check (Actual machining data is rewritten).
Δ: The value varies depending on the parameter setting.

■Foreground

Data type	Data status	Explanation
Parameters	×	The programmable parameter input commands which have been executed at the program check are set as actual data.
Local variables	Δ	It is possible to return to the status prior to starting the program check based on the value of the parameter "#1231 set03/bit0" (Graphic check compatibility parameter). 0: Returns to original status after check completion 1: Does not return to original status after check completion
Common variables	Δ	
Workpiece coordinate system offset	Δ	
Tool compensation amount	Δ	
Extended common variables	×	The extended common variable input commands which have been executed at the program check are set as actual data.

■Background

Data type	Data status	Explanation
Parameters	○	The programmable parameter input commands which have been executed at the program check are not set as actual data.
Local variables	○	"#1231 set03/bit0" (Graphic check compatibility parameter) is disabled.
Common variables	○	
Workpiece coordinate system offset	○	
Tool compensation amount	○	
Extended common variables	○	

4.1.18 Notes for Graphic Check (2D)

Check function

(1) The check function is valid for only NC internal operation results. Thus, the path is not drawn for the command which requires machine movement.

(2) For the handling of various types of data being set in the program, refer to "4.1.17 Handling of Variable Commands, Programmable Parameter Input, and Programmable Compensation Data Input".

(3) If the menu [M0/M1 neglect] is OFF, the program check is stopped by the M00/M01 command. The drawing is also stopped during M01 command execution regardless of the ON/OFF switch status of the optional stop switch signal (XC41).

(4) When the menu [M0/M1 neglect] is ON, the program check does not stop even if M00/M01 is commanded.

(5) "Program check completed" appears with M02/M30 command.

(6) Graphic check is not available for serial and MDI program.

(7) Check search is not available for the program whose file name is consist of 33 characters or more.

(8) When the check is executed, the drawing starts from the machine coordinate zero point. However, the machine coordinate zero point is (0, 0, 0) even when the value is set to the parameter "#2037 G53ofs" (zero point return).

(9) The drawing coordinate system settings of 2D and 3D graphic check are not linked.

(10) Drawing is continued even after changing to another screen during the tracing or checking modes.

(11) Graphic check (2D) cannot be performed during the trace mode.

(12) If the screen is switched to 2D check screen during trace, the confirmation message for trace OFF appears. Only when trace is turned OFF, the transition to 2D check screen is performed. If trace is not turned OFF, the message "Tracing" appears and 2D check screen displays blank.

(13) When trace window is displayed during check, check will be terminated. (This applies for all the part systems under checking.)

(14) The drawing is erased if the performance such as enlarge, reduce, move, change display mode, or rotate is set during check drawing. The program does not stop.

(15) When part systems are switched 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).

(16) Status during checking (in "Check continu"or "Check step") is held per part system.

(17) Check reset is available only for the displayed part system.

(18) When the drawing coordinate system is set for other than "All wk. Coord", the drawing is not executed because the drawing is performed in currently set coordinate system when there is no command to switch the coordinate system in the program to execute the program check. Make sure the coordinate system is specified in the program.

(19) When the parameter "#1231 set03(bit4)" (Switch zero point mark display position) is set to "1", and when the workpiece coordinate system or extended workpiece coordinate system is selected as the drawing coordinate system, the displayed zero point mark's position is not of the selected coordinate system to draw, but of the workpiece coordinate system or extended workpiece coordinate system specified by the machining program being checked.

(20) When the rotary axis (C axis) is specified by the parameter "#19405 Rotary ax drawing" and the C axis is used as the spindle position control (spindle C axis control) axis, a path corresponding to the rotary axis will be drawn as long as the rotary axis moves regardless of the state.

(21) The machine position of the rotary axis may not be temporarily clamped at 360°.

(22) When the rotary axis drawing function is ON, an arc in cylindrical interpolation will be drawn as a straight line.

(23) If Nose R compensation is carried out during cylindrical interpolation with the rotary axis drawing function ON, the tool path may be different from the programmed path.

(24) The check search cannot be executed when the sub part system that cannot be operated is displayed. If the search is carried out, the message "Search error" appears. (Only Lathe system)

(25) Set a different axis for each base axis parameter I, J or K (#1026 to #1028). If a same axis is set, the path might be drawn differently from the actual machining.

Precautions for drawing on foreground

(1) If you press the cycle start button when an arbitrary part system is being checked, the operation alarm "M01 program check mode" is generated and the cycle start is disabled while the button is held down.
Press the auto start button after the check is completed or after the check is stopped with the menu [Check reset].

(2) If you carry out the operation search or check operation (continuous or step check) during cycle start or automatic stop of an arbitrary part system, an operation message "Executing automatic operation" is displayed, and no search or check drawing is carried out.

(3) "Program check completed" appears with M02/M30 command. For the tape mode, the rewind process is not carried out.

(4) The check ends with the reset (NC reset or the menu [Check reset]).
When a program error occurs, cancel the error with the operation indicated above.
If NC reset is selected when the check screen is being displayed, check drawing ends but the menu to switch to the 2D check or 3D check is kept invalid. To enable this switching, press the menu [Check reset].

(5) During automatic operation, the contents of automatic operation program are displayed by the "Currently executed machining program" (ONB display) and "Buffer display" on the check screen. During checking, the contents of the checking program are displayed by the "Currently executed machining program" (ONB display) and "Buffer display" on the monitor screen.

(6) If you carry out the operation search or check operation (continuous or step check) when the program is restarted, the "Program restarting" operation message is displayed, and no search or check drawing is carried out.

(7) If the manual operation mode is selected during checking, the "M01 program check mode" is displayed and the axis cannot be moved. If you need to move the axis, temporarily stop the current check by executing a reset and then move the axis. When you switch from the manual operation mode to the memory mode, the alarm is released. You can continue the current check even if an alarm is output.

(8) When the PLC Interrupt signal is turned ON during checking, the "M01 Program check mode" is selected. To start the PLC interrupt, terminate the current check by executing a reset. When you turn the PLC interrupt signal OFF, the alarm is released. You can continue the current checking even if the PLC Interrupt signal is ON.

(9) The check mode ends with the reset or other operations, such as setting the tracing ON.

(10) All part systems are reset if NC reset is executed during checking. If a part system other than displayed part systems is being checked, switching the part system executes the program check from the top of the program.

(11) If the origin set operation is performed during checking, the result of the origin set is reflected to the value of the workpiece coordinate counter; however, the value returns to that before the check when the check is completed or with the reset.

Precautions for drawing on background

(1) The program that you have selected during Check Search is not the program that is executed during automatic operation. Search the program for automatic operation by using the operation search function of the monitor screen.

(2) The parameters and compensation amount do not reflect on check drawing even if you set them during checking. To reflect them on the drawings, reset the check once. Then, press the menu [Check continu] or [Check step] and execute the check again.

(3) "Program check completed" appears with M02/M30 command.

(4) The check is not terminated via execution of a reset.
If a program error occurs, release the error by selecting the menu [Check reset].

(5) When part systems are switched during checking, drawing will be erased. Note that even if the part system is returned to the original one, its drawing during cutting is not displayed.

(6) During check search, if the length of the full path name exceeds 48 characters, you cannot move to another directory.

(7) If the menu [Optional Skip] is ON (highlighted), the blocks that have the "/" character are skipped and the T-code list is obtained.

(8) The file name continues to be displayed as an O-number display even if you erase or rename the file after it is checked and searched. If you check a file in this status, the "Push [Check reset] menu" message is displayed.

(9) During checking, if you read the integrated time by using variable number #3001 or #3002, the checking start time is obtained. You cannot write the integrated time.

(10) When [Check continu] is executed, even if variable number #3006 is used for checking, you cannot stop the program and you cannot display the message.

(11) The check mode ends with the operations, such as setting the tracing ON.

Full screen display of program check function (2D)

(1) This function is an optional specification. When the menu [M0/M1 neglect] is set to OFF and trace 2D check is enabled, the graphic drawing area is enlarged or reduced horizontally or vertically if the normal display is changed to the full-screen display. At this time, the scale value remains unchanged.

Precautions for automatic graphic

(1) Auto graphic is executed only for the 1-plane display mode. The menu [Auto Graphic] cannot be selected for the other modes. The menu is grayed out. Auto graphic mode always remains as it is in the 1-plane display mode.

(2) Once the auto graphic mode is set, it is retained after the power is turned ON again.

(3) The auto graphic cannot be set for the trace function.

(4) The auto graphic mode is common between part systems.

(5) If the display size is switched (between normal and full screen) or the display range is changed during program check, the auto graphic drawing is not carried out even if the auto graphic is valid.

(6) If a program error occurs during the auto adjustment of the scale, a message "Push [Check reset] menu" appears, and no auto adjustment of the scale or graphic drawing is carried out.

(7) Even the M00 or M01 command is issued, the program does not stop.

(8) If a program which contains M99 command is selected, auto adjustment of the scale will be executed repeatedly and the adjustment will not be completed with the "Executing auto scale" message blinking.
Do not select a program which contains M99.
Execute check reset when it happens.

(9) When a large capacity program is selected, it may take time to perform the auto graphic.

(10) If the display position of a figure is changed by changing the display range, auto graphic drawing is not carried out.

Precautions for grid display

(1) Grids are displayed only in the 1-plane display mode. The menu [Grid display] cannot be selected in any mode other than the 1-plane display mode. The menu is grayed out. During the grid display mode, the state in the 1-plane display mode is maintained.

(2) If the display mode is switched (to 2-plane or 3-dimensional) while the grid display is valid, the scale extension line is erased. However, the scale extension line is displayed when the mode is returned to the 1-plane display mode.

(3) The set grid display is maintained even after the screen is switched. However, it is not maintained after the power is turned OFF and ON.

(4) The grid display mode cannot be set for the trace function.

(5) The grid display mode is common between part systems.

Graphic check rotary axis drawing

When the parameter "#19405 Rotary ax drawing" is set to "0" (disabled) in the program check function (2D), and a rotary axis is designated with any of the base axis parameter I, J or K (#1026 to #1028) due to the machine configuration, the movement of the C axis is drawn with a linear path. Setting the base axis parameter for graphics I, J or K (#8631 to #8633) enables the C axis not to move with linear path. Set the value as described later to the base axis parameter for graphics corresponding to the base axis parameter in which the C axis is designated; the name of the axis which is not used for the base axis parameter among the names of the linear axes that configure the plane (except for the name of the C axis). Set the names of the linear axes, that configure the plane, in the other base axis parameters for graphics.
Select the plane or cube on which the graphic is to be drawn by the menu [Display mode], and the linear path drawing due to the C axis movement is not performed at this time.

The setting of the base axis parameter for graphics is valid when the following conditions are satisfied.

If the conditions are not satisfied, the setting of the base axis parameter for graphics is ignored and the graphic is drawn by the setting of the base axis parameter.

[Enabling conditions for the base axis parameter for graphics]

- The specification of "Graphic check rotary axis drawing" is invalid.
- The C axis is designated in the base axis parameter I, J or K (#1026 to #1028).
- The axis name which is not used for the C axis is set in the base axis parameter for graphics corresponding to the base axis parameter in which the C axis is designated. (When the C axis is designated as an axis name, the linear path drawing due to the C axis movement is performed.)

Note

(1) When the setting of the base axis parameter for graphics has been changed (including when no value is set), the new setting is enabled after you re-select the plane or cube by the menu [Display mode] on the 2D check screen. Also, when the power is turned ON, the setting is enabled after re-selecting the menu [Display mode].

Setting example of the base axis parameter for graphics

Axis configuration (#1013)	Base axes I, J and K (#1026 to #1028)	Base axes for graphics I, J and K (#8631 to #8633)	OK or NG for the setting
[X, Z, C]	[X, C, Z]	[X, Y, Z], [Z, Y, X]	OK
		e.g. [X, Z, Y], [Z, X, Y]	NG
[X, Y, C]	[X, Y, C]	[X, Y, Z], [Y, X, Z]	OK
		e.g [X, Z, Y], [Y, Z, X]	NG
[Y, Z, C]	[C, Y, Z]	[X, Y, Z], [X, Z, Y]	OK
		e.g. [Y, X, Z], [Z, X, Y]	NG

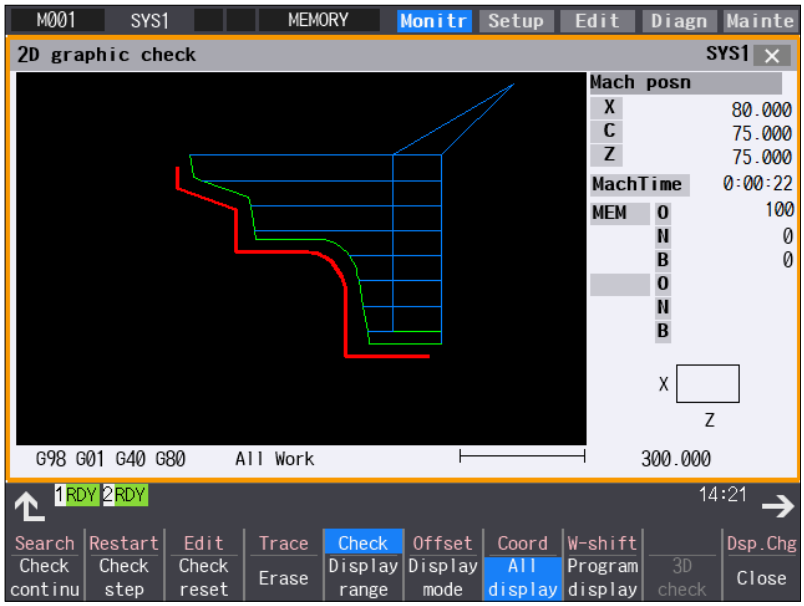
If the setting value is in NG state, you may not select the plane or cube by the menu [Display mode] on the 2D check screen.

Finish shape view before compensation of tool shape for turning machining (Compound type fixed cycle for turning machining)

The tool shape compensation for turning machining of compound type fixed cycle for turning machining draws a different machining path from the finish shape commanded with the machining program. Because it compensates the finish shape itself.

Finish shape view before compensation draws the path of the finish shape before compensation when any of G71 to G73 is commanded in the graphic check (2D). The finish shape view before compensation is displayed, then the finish shapes before and after compensation can be compared by displaying the rough cutting cycle with G71 to G73 in the normal manner. That enables to check the uncut section left by tool shape compensation for turning machining and the change of finish shape.

To display the finish shape view before compensation, set the parameter "#8137 T. tool shape comp" to "1" and "#8138 Pre-compens. Shape" to "0".



Note

(1) While the finish shape before compensation is being drawn, the machining time is not counted.

(2) The finish shape view before compensation is displayed only in the graphic check (2D). This is not displayed in other display functions (such as graphic trace and graphic check (3D)).

(3) For the finish shape before compensation, the compensation function other than the tool shape compensation for turning machining such as nose R compensation is enabled.

(4) In the finish shape view before compensation, the program can also be drawn by one block with [Check step] menu.

(5) In the finish shape view before compensation, the following movements are not drawn.
- from the cycle command point to the finish shape start block
- from the finish shape end block to the cycle command point

4.2 Graphic Check (3D) (M System)

Edit – 

Graphic check (3D) is a function that allows you to check the machining program movement path only by drawing without automatic operations. Therefore, the graphics can be drawn at a high speed and high accuracy, and the machining program can be checked.
Press the menu [Check] on the monitor or edit screen to switch to the graphic check screen.
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.

This function requires workpiece and tool shape settings.

Using the menu [All display], normal display and whole display modes can be switched.

Note

(1) When changing the screen to the edit screen, display depends on the previous checking type (2D/3D). If 2D graphic data appears, press the menu [3D check]. Note that when transferring to this screen during graphic trace, 3D solid program check is displayed.

[Terms]

The following descriptions are the meanings of the terms used in this chapter.

Foreground: Active state operated by the user such as automatic operation on the NC

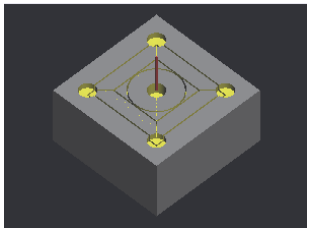
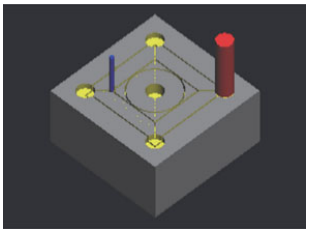
Background: Inactive state operated in the back of foreground

[Check method]

There are two types of 3D graphic check as below, which can be switched by [Type change] menu.

Note

(1) Make sure to select the check method before the check is started. During the check, [Type change] menu is grayed out, and the check method cannot be changed.

Check method	Outline of drawing	Details
Check method I	This method simplifies the machining result of machining program and simulates at a high speed. A machining program in single part system (displayed part system) is drawn. In 3D drawing area, one workpiece and one tool are displayed. 	This method is operated on the background on M800VW Series, M800VS Series and M80VW Series. This method is operated on the foreground on M80V Series.
Check method II	This method simulates the machining result of machining program at a higher accuracy than check method I. Machining programs in multiple part systems can be drawn simultaneously. In 3D drawing area, one workpiece and tools for part systems are displayed. 	This method is available only when the operation mode in the part system to be simulated is in the memory mode. This method is operated on the foreground and in the machine lock status during the program check. Also, during the program check, the PLC signal (Graphic check (Check method II) in progress X759) is turned ON. The program check cannot be started during automatic operation in arbitrary part system. The program check is in operation from the program execution by pressing the menu [Check continu] or [Check step] on the check screen to the following status. •Program ends. •The menu [Check reset] is pressed. •The reset signal or emergency stop signal outputs. •The screen is moved from the check screen to another one.

The following functions are supported with check method I and check method II.

○: Available ×: Not available Δ: Depending on the model

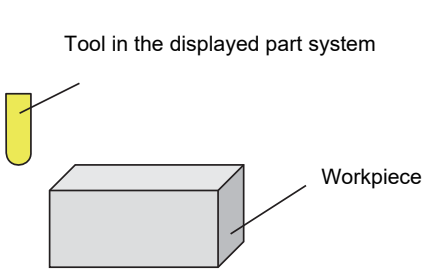
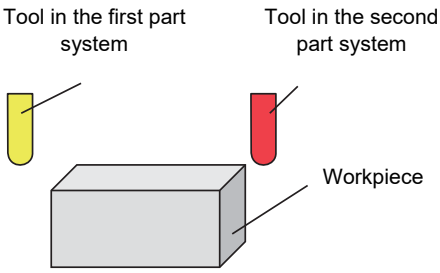
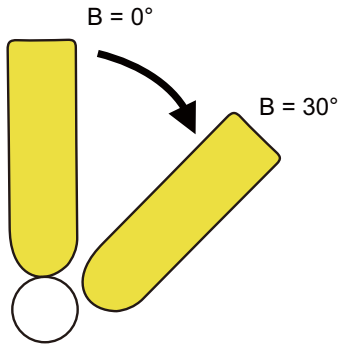
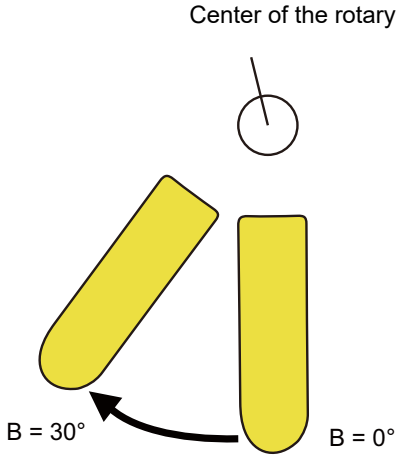
Item	Check method I	Check method II
Operation in a single part system	○	○
Multi-part system simultaneous operation	×	○
Coordinate system selection	Δ (*3)	○
Tool center point control	×	○
Inclined surface machining	○	○
Travel command of the rotary axis on tool-side	○	×
Operate on the background	Δ (*1)	× (*2)

(*1) Operation on the background is available on M800VW Series, M800VS Series and M80VW Series, but operation on the foreground is available on M80V Series.

(*2) With check method II, operation only on the foreground is available regardless of the models. During the check, the high-speed simple program check is started. For the limitations of operation on the foreground, refer to "4.2.17 Notes for Graphic Check (3D)".

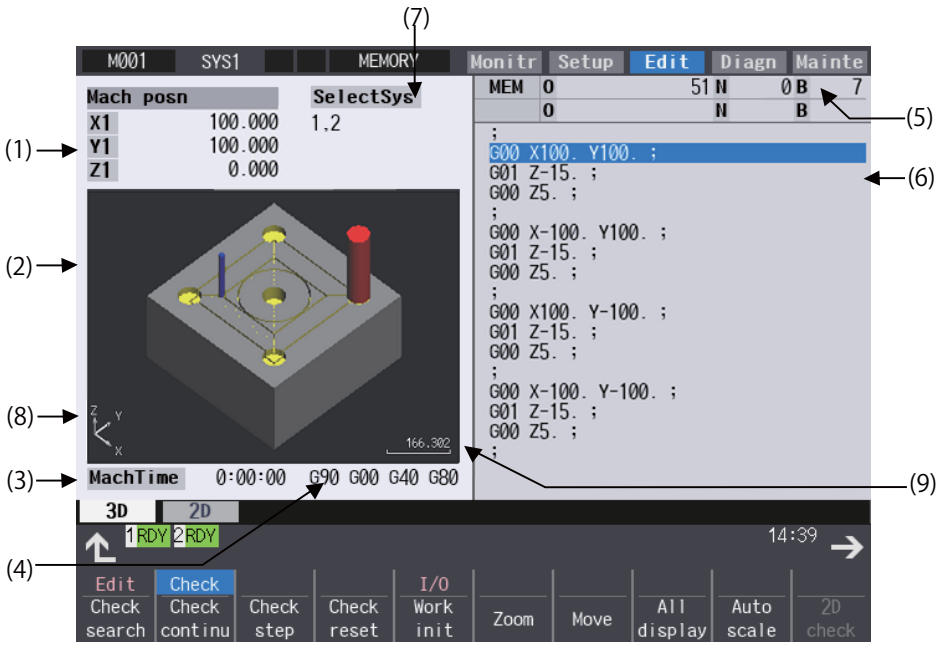
(*3) Movement of the rotary axis on tool-side differs between check method I and check method II. For details, refer to the following table.

The following shows the differences of drawing contents between check method I and check method II.

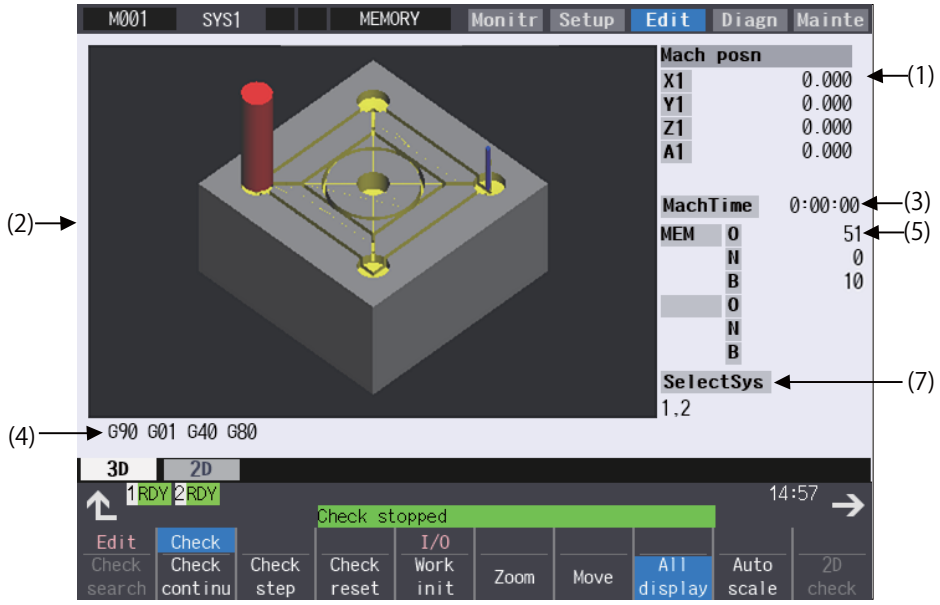
<table>
<tr><th>Item</th><th>Check method I</th><th>Check method II</th></tr>
<tr><td>Multi-part system simultaneous operation</td><td>Only machining result of the machining program in the displayed part system is drawn (One tool is displayed).

</td><td>Machining results of the machining program in multi-part system are drawn simultaneously (Tools are displayed for the number of the part systems).

</td></tr>
<tr><td rowspan="2">Travel command of the rotary axis on tool-side</td><td>When the command for the rotary axis is issued, tool rotates around its nose so that the graphic is not drawn correctly.

</td><td>When the command for the rotary axis is issued, tool rotates around the center of the axis.

</td></tr>
<tr><td colspan="2">For details on the drawing of rotary axis travel command, refer to "4.2.14 Drawing Travel Command of the Rotary Axis".</td></tr>
</table>

Item	Check method I	Check method II
Coordinate system used for 3D drawing	You can select either machine coordinate system or workpiece coordinate system. For details, refer to "4.2.10 Setting the Workpiece Shape".	Machine coordinate system only.

■Normal display ([All display] is OFF)



■Full-screen display ([All display] is ON)



Display items

Display items			Details
(1)	Check counter		This displays the axis counter for the 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. In the normal display mode, the axis names common to part systems ("#1022 axname2") based on the basic axis name set by the parameters "#1026 base_I", "#1027 base_J", and "#1028 base_K" are displayed. In the full-screen display mode, maximum 5 axes are displayed. Three axes are set by #1026,#1027 and #1028, and two axes are set by the rotary axis configuration parameters. If no axes are set for the rotation axis configuration parameter, the 2 rotation axes set in parameter "#1013 axname" will display in sequence. However, when the specific parameters are set without setting the rotary axis configuration parameters, two rotary axes including one set in "#1516 mill_ax" out of those set in "#1013 axname" are displayed. The specific parameters are listed in the table in the "Drawing travel command without setting the rotary axis configuration parameters" section in "4.2.14 Drawing Travel Command of the Rotary Axis". [Check method I] This displays the workpiece coordinate counter for the current part system (the part system being checked). [Check method II] This displays the machine coordinate counter for the current part system (it is not necessarily the part system being checked).
(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		The time required for machining is calculated and displayed.
(4)	G modal display for check		The following items are displayed from the left. ♦Absolute/incremental (G90, G91: Group 3) ♦Operation mode (G00, G01, etc.: Group 1) ♦Tool compensation (G40, G41, G42: Group 7) ♦Fixed cycle (G70 etc.: Group 9) The items to be displayed vary depending on the status of group 24 command modal.
(5)	Currently executed machining program	Main O10...	This displays the device name, program No., sequence No., and block No. currently being executed. When they exceed the displayable number of characters (12 characters), the excess is abbreviated and displayed (*).
		Sub O1234…	This displays the sub-program device name, program number, sequence number, and block number for the sub-program currently being executed. When they exceed the displayable number of characters (12 characters), the excess is abbreviated and displayed (*).
(6)	Buffer display		This displays the contents of the machining program currently being executed. The block being executed is highlighted. When the parameter "#19431 Num display in prg" is set to "1", the variable, operation or function in the numerical part that follows each address is replaced with the calculated value and displayed for the current block and the next block.
(7)	Part system to be checked (Check method II only)		This displays the part system to be checked (the part system for which the parameters "#8982 CheckSmltns$1Invld" to "#8989 CheckSmltns$8Invld" are set) (up to eight part systems). This menu is displayed with check method II only.
(8)	Coordinate system display		This displays the coordinate system direction. This displays the axis direction of the base axis I/J/K with arrow.
(9)	Scale display		This shows the display range scale. The unit is set to "mm/80pixcel" for metric system and "inch/80pixcel" for inch system.

Menus

Menus	Details
Check search	This performs a check search. Select a program to be checked from the program file list displayed in the pop-up window.
Check continu	This checks the program continuously.
Check step	This checks the program per block.
Check reset	This resets 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).
Zoom	This enlarges/reduces the displayed workpiece shape.
Move	This moves the displayed workpiece shape.
All display	This switches the normal display mode and the full-screen display mode.
Auto scale	The scale is automatically set so that the unmachined workpiece shape set at the "Work set" window (workpiece shape setting) to be approximately 90% of the graphic area.
2D check	This changes the program check (2D) display. This menu cannot be selected while the check is performed or interrupted.
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.
Rotate	This rotates the displayed workpiece shape.
Regist. Image	This associates a graphic check image with a machining program and save the program.
Scale display	This displays the scale of coordinate system direction and display range in the graphic drawing area.
M0/M1 neglect	[Check method I] If [M0/M1 neglect] is ON, the program check does not stop even when M00/M01 is commanded. If [M0/M1 neglect] is OFF, the program check stops when M00/M01 is commanded. [Check method II] The menu [M0/M1 neglect] is grayed out and non-selectable. The program check does not stop in the block where M00/M01 is commanded. (*1)
Type change	This switches between check method I and check method II. <Note> ♦When check method switch is invalid (parameter "#1231 set03/bit7" is set to "1"), this menu is not displayed.
Work setting	This sets the workpiece shape used at the solid display.
Tool setting	This sets the tool shape used at the solid display.

(*1) M code independent output (DM00/DM01) follows the parameter setting indicated below.
- #1284 ext20/bit3 (M code output during multi-system simultaneous simulation (Check type II))
- #1405 M_mode(SMLK) (M code output (during high-speed simple program check)
- #1451 M[M031-000](SMLK) to #1466 M[M511-480](SMLK) (Special operation registration M code (High-speed simple program check))

Note

(1) When automatic operation is running during the graphic trace mode, the menu [Check search], [Check continu], [Check step], [Check reset] cannot be selected and the 3D solid program check cannot be performed. If the program is not in the process of automatic operation, a check is possible only for 3D solid.

(2) The menus [2D check], [Check search], [Work setting] and [Tool setting] cannot be selected while the check is performed or interrupted.

(3) The menus [M0/M1 neglect] and [Regist. Image] are grayed out during continuous check, and the menus cannot be selected.

(4) The coordinate system direction and the scale display are held after the screen is switched, but they are not held after the power is turned OFF and ON again.

Sub-menus of [Check search]

Menus	Details
Memory	This selects the device for searching for the program. When a device with directory is selected, the root is selected first.
Memory2	
HD	
Memory card	
DS	
USB Memory	
Top jump	Display starts from the first page of the directory/list of files. The cursor moves to the top line.
Bottom jump	The last page of the directory/list of files is displayed. The cursor moves to the bottom line.
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.
Preview	This switches to show or hide the preview (detailed information).
Change Disp	This switches between list display mode and thumbnail display mode.
List update	This updates the list contents. (The latest contents of the currently selected device and directory are listed.)
Sort change	This changes the method of sorting the list.
Close	This closes the pop-up window and quits this function.

4.2.1 Switching Check Method

Press the menu [Type change] to switch between check method I and check method II. Every time you press the menu, the type is switched between check method I and check method II.
The menu [Type change] is displayed when the check method switch is valid (parameter "#1231 set03/bit7" is set to "0").

You can confirm whether it is in check method I or II by the indication of "SelectSys" on the screen.

Check method I: "SelectSys" is not indicated.

Check method II: "SelectSys" is indicated.

4.2.2 Checking Continuously

There are three types of method for checking the program.

- Checking the specified program
- Checking the program currently opened on the edit screen (Edit)
- Checking the program searched on the monitor screen (Monitr)

Operation method (Selecting a program to be checked)

Checking the specified program

(1) Press the main menu [Check] on the edit screen.
Press the menu [3D Check] when the program check (2D) appears.

(2) Press the menu [Check search].
If you press the menu [Comment nondisp] in the second page, the comment is hidden.

(3) Select the device.
(Example) Menu [Memory]

→ The selected device name and directory (memory: /program) appear in the device name and directory display columns.
For the devices other than NC memory or NC memory 2, root directory is selected.

(4) (For the devices other than NC memory or NC memory 2)
Use the [↑], [↓], [PAGE↑], or [PAGE↓] key to align the cursor with a directory of the program.

Press the [INPUT] key to move to inside the directory.

<Note>

- If the list contents differ from the actual device or directory, press the menu [List Update].
- Press the menu [Sort change] to change the sort. Each time the menu key is pressed, the sort method shifts in the following order: file names in ascending order -> file names in descending order -> dates and comments in ascending order -> dates and comments in descending order -> file names in ascending order.

(5) Press the [↑], [↓], [PAGE] or [PAGE] key to move the cursor to the program to be searched.
The name of the machining program to be edited can be input in the input area.
When the touchscreen display unit is used, you can also select a program by touch operation.

(6) Press the [INPUT] key.
When setting data exists in the input area, a search is performed in this data. This procedure is also used when performing an NB search.
(Example)
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

→ The check search is started.
When the search is completed, the operation message "Search completed" is displayed.
The searched device and program position appear in the display area for the machining program currently being executed.
The list display is closed.
The system distinguishes between presence and absence of an extension, and therefore due care should be taken.

Note

(1) The operation message "Sorting of tool data is in progress" is displayed when tool data is being sorted, and the check search is disabled.

(2) When you perform an operation search by designating a block No., it can be set by the parameter "#1760 cfgPR10/bit5" whether the block which consists of comment only is counted as one block. When the parameter is set to "0", the comment block is counted as one block.
Set the parameter to "1" to count the comment block as same as the automatic operation.

(Example) When an operation search is performed by the sequence No. 1 and the block No. 2

Sequence Nos. and the block Nos. vary as indicated in the following table depending on the setting of the parameter. Thus, the different block is searched.

When "#1760 cfgPR10/bit5" is set to "0", the block (a) is searched.

When "#1760 cfgPR10/bit5" is set to "1", the block (b) is searched.

Program	#1760 cfgPR10/bit5 = 0		#1760 cfgPR10/bit5 = 1	
	Sequence No.	Block No.	Sequence No.	Block No.
N01;	N1	B0	N1	B0
(COMMENT1);	N1	B1	-	-
G28 X0Y0Z0; .....(a)	N1	B2	N1	B1
G0 X10.; .....(b)	N1	B3	N1	B2
M30;	N1	B4	N1	B3

Checking the program currently opened on the edit screen

(1) Press the menu [Program check] on the edit screen.
Press the menu [3D Check] when the program check (2D) appears.

Note

(1) Regardless of the check method or the setting of the parameters "#8982 CheckSmltns$1Invld" to "#8989 CheckSmltns$8Invld", the program opened on the edit screen is drawn as a machining program in single part system (displayed part system). Only the displayed part system is displayed in "SelectSys" on the screen.

Checking the program searched on the monitor screen

(1) Press the menu [Check] on the edit screen.
Press the menu [3D Check] when the program check (2D) appears.

Note

(1) With check method II, among the part systems to be checked (parameters "#8982 CheckSmltns$1Invld" to "#8989 CheckSmltns$8Invld"), the part system being searched on the monitor screen is drawn.

Operation method (Executing the program check)

(1) Press the menu [Check continu].

Behaviors during check operation

- Press the menu [Check step], the program check is carried out block by block starting from the next block.
- Press the menu [Check continue], the program check is stopped. Press the menu [Check continu] during block stoppage, the program check is carried out continuously.
- Press the menu [Check reset], the program check is reset after completion of the currently executing block.



The program check is executed for one block, and workpiece shape in the cutting process is drawn.
A "Checking" message appears and the menu [Check continu] is highlighted.
The check counter display is updated.
The check G modal is not displayed.
The machining time is updated.

When the check is completed, a message appears and the mode returns to normal.
The menu highlight returns to normal.

Note

(1) The operation message "Sorting of tool data is in progress" is displayed when tool data is being sorted, and the check is disabled.

4.2.3 Checking One Block at a Time

There are three types of method for checking the program.

- Checking the specified program
- Checking the program currently opened on the edit screen (Edit)
- Checking the program searched on the monitor screen (Monitr)

Perform the same procedure as "4.2.2 Checking Continuously" on selecting a program.

Operation method (Executing the program check)

(1)	Press the menu [Check step].	The program check is executed for one block, and workpiece shape in the cutting process is drawn. The check counter display is updated. The check G modal display is updated. The machining time is updated.
(2)	Press the menu [Check step] again. Behaviors during check operation • Press the menu [Check continue], program check is carried out continuously up to the end of the program. • Press the menu [Check reset], the program check is reset.	The program check is executed for the single block. When the check is completed, a message appears and the mode returns to normal. The menu highlight returns to normal.

Note

(1) The operation message "Sorting of tool data is in progress" is displayed when tool data is being sorted, and the check search is disabled.

4.2.4 Checking Multiple Part Systems Simultaneously

With check method II, machining programs in multiple part systems can be drawn simultaneously. If you press the menu [Check continu] or [Check step], the movement path of the machining program in the part system being searched is drawn among the target part systems. Select the target part system with the parameters "#8982 CheckSmltns$1Invld" and "#8983 CheckSmltns$2Invld" (The part system which the parameter is set to "1" is out of the target).

Operation method (Selecting a program to be checked)

(1) Press the main menu [Check] on the edit screen.
Press the menu [3D Check] when the program check (2D) appears.

(2) Press the $\leftrightarrow$ key to switch to the part system to be checked.
In addition to the above, the PLC signal and touch-screen operation are provided as the part system switching method.

(3) Press the menu [Check search].
For details on how to search the machining program, refer to "4.2.2 Checking Continuously".

(4) Repeat (2) and (3) to check the machining program of the part system to be checked.

Note

(1) When the multi-part system program management is valid (#1285 ext21/bit0 =1, ext21/bit1 =1), since the check search is executed across all part systems in batch only for the machining programs in NC memory or NC memory 2, the step (4) is not necessary.

Operation method (Executing the program check)

(1) Press the menu [Check continu] or [Check step].
For details on how to execute the check, refer to "4.2.2 Checking Continuously" and "4.2.3 Checking One Block at a Time".
➡ The program check is executed, and workpiece shape in the cutting process is drawn.

Note

(1) If you press the menu [Check step], the check is executed (is drawn) on the program of displayed part system by one block. The process of the non-displayed part system moves ahead by the time of one block check for the displayed part system. Thus, the drawing of the non-displayed part system may be interrupted in the middle of a block, or it may proceed for multiple blocks.
The program check is not executed on the part system for which the parameter (any of #8982 to #8989) is set to "1" (check-invalid part system).
When the displayed part system is set as a check-invalid part system, the check is executed on the part system whose No. is the smallest among the check-valid part systems by one block.

(2) If any part system to be checked is not set, the operation message "No check system" is displayed.

4.2.5 Canceling the Program Check

Operation method

(1) Press the menu [Check reset]. → The check mode is released, and the menu highlight returns to normal.
The operation message "Reset complete" appears.

Even when the screen transits from the check screen during the program check, the program check is canceled.

With check method II and check method I (M80), the program check is canceled in the middle if the reset signal or the emergency stop signal outputs.

4.2.6 Enlarging and Reducing the Workpiece Shape

Operation method

(1) Press the menu [Zoom].

(2) Press the menu [Zoom in].
To enlarge the figure:
Press the menu [Zoom in] or [-] key.
To reduce the figure:
Press the menu [Zoom out] or [+] key.

4.2.7 Moving the Workpiece Shape

Operation method

(1) Press the menu [Move].

(2) Use the menu [Up], [Down], [Left], or [Right] to adjust the display position.
The cursor movement keys ([↑], [↓], [←], [→]) can also be used for moving.

4.2.8 Rotating the Workpiece Shape

Operation method

(1) Press the menu [Rotate].

(2) Use the menu [Up], [Down], [Left], or [Right] to adjust the display angle.
The cursor movement keys ([↑], [↓], [←], [→]) can also be used for moving.

Note

(1) The set display angle is maintained even after rebooting the machine.

(2) When all of storage data concerning the drawing such as display angle become zero by NC data reset, the scale setting is automatically performed (same state as when the menu [Auto scale] is pressed).

4.2.9 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.

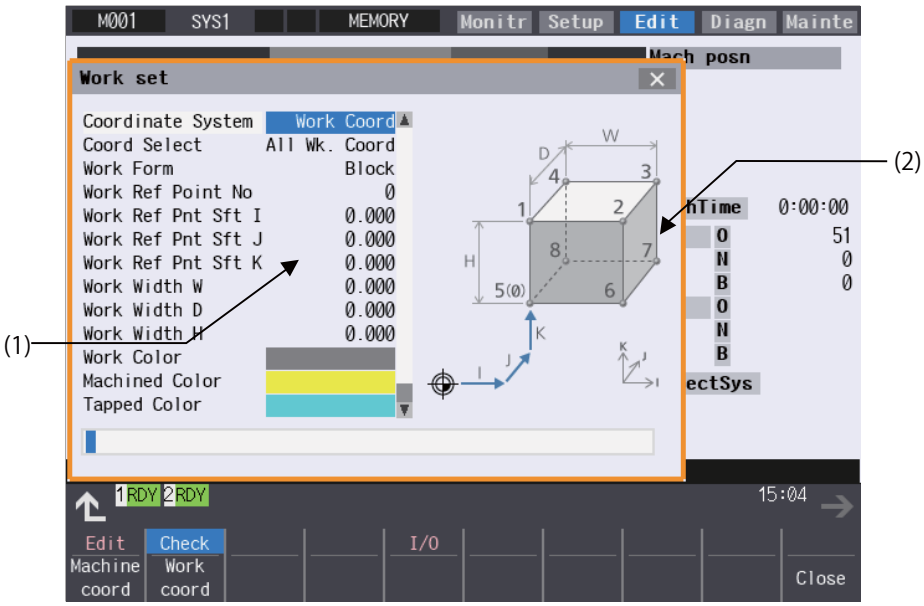
Operation method

(1) Press the menu [Colli check].

(2) Press the menu [Check step] or [Check continu]. → The program check is executed for one block, and workpiece shape in the cutting process is drawn.
A "Checking" message is displayed and the menu [Check step] or [Check continu] is highlighted.
The workpiece and tool contact, this is drawn in a color highlighting the interference.

4.2.10 Setting the Workpiece Shape

Press the menu [Work setting] to display a pop-up window similar to the following, and set the workpiece shape used in the solid display.



Display items

Display items		Description
(1)	Workpiece shape setting area	This sets all workpiece shape items.
(2)	Workpiece setting 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 "Coordinate System" (drawing coordinate system))

Menus	Description
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).
Work coord	
Close	This closes the pop-up window and quits this function.

Menus (the cursor is at "Coord Select" (workpiece coordinate system selection))

Menus	Description
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"
Coord G54-G59	This specifies the workpiece coordinate (G54 to 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.
Coord G54.1 P_	This specifies the external workpiece coordinate (G54.1 Pn) to the drawn coordinate system type. The extended workpiece coordinate (G54.1 Pn) can be selected for the type of coordinate system to draw by inputting the P No. If the drawing coordinate system is "Machine coord", this is displayed in gray and cannot be selected.

Menus	Description
Close	This closes the pop-up window and quits this function.

Menus (the cursor is at "Work Form" (material shape))

Menus	Description
Default shape	This sets the material shape. The drawings are made as a block when the default shape is selected.
Block	
Close	This closes the pop-up window and quits this function.

Menu (the cursor is at "Work Ref Point" (material basic point), or "Work Width" (material width))

Menus	Description
Close	This closes the pop-up window and quits this function.

Menus (the cursor is at the "Work Color", "Machined Color", "Tapped Color", or "Interference Color")

Menus	Description
Default color Gray Red Yellow Blue Green Light blue Purple Pink	This sets the material color (Work Color), machining surface color (Machined Color), tap surface color (Tapped Color), interference surface color (Interference Color). 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 <Note> ♦The "Tapped Color", that is the same as when the tapping tool is selected, is used on the "Work set" screen. Therefore, switching of the tap surface color and normal surface color depends on not G command but the selected tool. However, even with the tapping tool being selected, if it is moved in the arc or helical direction, the "Work set" screen does not have the "Tapped Color".
Close	This closes the pop-up window and quits this function.

Setting the drawing coordinate system

(1) Use the [↑] or [↓] key to move the cursor to the drawing coordinate settings column.

(2) Press the menu [Work coord].
It is also possible to set input values.
0 [INPUT]: Machine coord
1 [INPUT]: Work coord

➡ "Work coord" appears at the cursor position.
The cursor moves to the next line.
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, the menu [Coord G54-G59] or [Coord G54.1P] cannot be selected.

(Example) When "Coord G54-G59" is selected

(1) Use the [↑] or [↓] key to move the cursor to "Coord Select" column.

(2) Press the menu [Coord G54-G59].

➡ The sub-menus, which select G54 to G59, are displayed.

(3) Press the menu [G56].
When the menu [Return] is selected, return to the workpiece coordinate system selection menus.

➡ "G56" appears at the cursor position.
The cursor moves to the next line.
The sub-menus of the material shape (WorkForm) are displayed.

Note

(1) The drawing coordinate system settings of 2D and 3D graphic check are not linked.

Setting the material shape (Work Form)

(1) Use the [↑] or [↓] key to move the cursor to the material shape settings column.

(2) Press the menu [Block].
It is also possible to set input values.
0 [INPUT]: Default shape
1 [INPUT]: Block

➡ "Block" appears at the cursor position.
The cursor moves to the next line.
A block guide drawing appears in the guide drawing area.

Selecting material zero point No.

(1) Use the [↑] or [↓] key to move the cursor to the setting column of "Work Ref Point No".

(2) Input "5" and press the [INPUT] key. → "5" appears at the cursor position. The cursor moves to the next line.

Input the value to set.
0 [INPUT]: Material zero point 0
1 [INPUT]: Material zero point 1
2 [INPUT]: Material zero point 2
3 [INPUT]: Material zero point 3
4 [INPUT]: Material zero point 4
5 [INPUT]: Material zero point 5
6 [INPUT]: Material zero point 6
7 [INPUT]: Material zero point 7
8 [INPUT]: Material zero point 8

Note

(1) Set the value in the range of 0 to 8 in "Work Ref Point No". If you set "0" to the "Work Ref Point No", the behavior is the same as the one when the zero point "5" is selected.

(2) If a value outside of the setting range is input, the operation message "Setting error" is displayed not to be set.

Setting the material zero point shift (I coordinates)

Set the shift amount between the zero point of the coordinate system and zero point of the workpiece. The shift amount to be set differs as follows, depending on the type of the coordinate system selected on the workpiece shape setting screen.
When the basic machine coordinate system is selected, set the shift amount from the machine coordinate zero point.
When the workpiece coordinate system is selected, set the shift amount from the machine coordinate zero point.

(1) Use the [↑] or [↓] key to move the cursor to the setting column of "Work Ref Pnt Sft I".

(2) Input data. → "100.000" appears at the cursor position. The cursor moves to the next line.
(Example) 100 [INPUT]

(3) Set the data for the other zero points and the widths of the workpiece in the same way.

Note

(1) The setting ranges of the material zero point are as follows.

#1003 Input setup unit	#1041 Initial 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

(2) If the position of the workpiece drawing zero point (material zero point) exceeds the above setting range due to the setting values of the material zero point shift and the width, the workpiece drawing zero point is converted to the maximum or minimum value of the setting range.

Setting the material width (in I axis direction)

Set the material width in millimeters in the millimeter system, and in inches in the inch system.

(1) Use the [↑] or [↓] key to move the cursor to the setting column of "Work With I".

(2) Input data.
(Example) 100 [INPUT]
→ "100.000" appears at the cursor position. The cursor moves to the next line.

(3) Set the other width data in the same way.

Note

(1) If "0" or a value outside of the range is set for the material width, the workpiece of the default size (width I = 100.000, width J = 100.000, width K = 50.000) is displayed.

(2) The setting ranges of the material width are as follows:

#1003 Input setup unit	#1041 Initial inch	
	0 (mm)	1 (inch)
B	0 to 99999.999	0 to 9999.9999
C	0 to 99999.9999	0 to 9999.99999
D	0 to 99999.99999	0 to 9999.999999
E	0 to 99999.999999	0 to 9999.9999999

(3) If the position of the workpiece drawing zero point (material zero point) exceeds the above setting range due to the setting values of the material zero point shift and the width, the workpiece drawing zero point is converted to the maximum or minimum value of the setting range.

Setting the material color (WorkColor)

(1) Use the [↑] or [↓] key to move the cursor to the material color setting column.

(2) Press the material color menu.
(Example) Menu [Gray]
→ The cell where the cursor is located appears in gray. The cursor moves to the next line. (The cursor does not move when at the interference color.)

It is also possible to set input values.
0 [INPUT]: Default shape
1 [INPUT]: Gray
2 [INPUT]: Red
3 [INPUT]: Yellow
4 [INPUT]: Blue
5 [INPUT]: Green
6 [INPUT]: Light blue
7 [INPUT]: Purple
8 [INPUT]: Pink

(3) Set the other color data in the same way.

<Note>

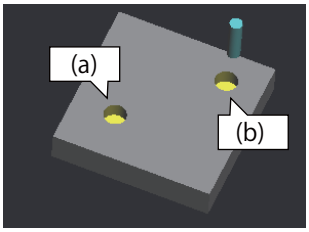
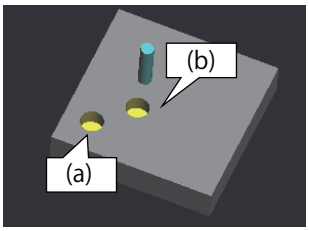
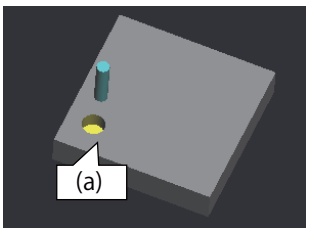
- The following colors are used if the default colors are designated.
  Material color: Gray
  Machining surface color: Yellow
  Tap machining surface color: Light blue
  Interference surface color: Red

Difference of drawing depending on the combination of the menus for "Coordinate System" and "Coord Select"

The drawing of 3D check changes as follows depending on the combination of the selected menus when the cursor is on "Coordinate System" and "Coord Select"

(Example)

[Parameter]	[Machining program]
#8920= 1 [Tool compensation] No.1: Length 150.000 L wear 0.000 Radius 20.000 R wear 0.000 [Workpiece coordinate system] G59: X 50.000 Y 50.000 Z 0.000 G54.1P1: X 200.000 Y 200.000 Z 0.000 [Workpiece shape setting] Work Ref Point (I,J,K) = (0.000, 0.000, -200.000) Work Width (I,J,K) = (500.000, 500.000, 100.000)	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 ; % ;

Coordinate system	Coord Select	Drawing result when 3D check in example	
Machine coord	All wk. coord	 (a) Drawing by G59 (b) Drawing by G54.1P1	Arranges the workpiece based on the machine coordinate zero point. Both workpiece coordinate systems G59 and G54.1P1 are drawn.
Work coord	All wk. coord	 (a) Drawing by G59 (b) Drawing by G54.1P1	Arranges the workpiece based on the workpiece coordinate zero point which is selected at the check execution. Both workpiece coordinate systems G59 and G54.1P1 are drawn.
Work coord	G59	 (a) Drawing by G59	Arranges the workpiece based on the workpiece coordinate zero point which is selected at the check execution. Only draws the workpiece coordinate system G59. The workpiece coordinate systems G54.1P1 is not drawn.

4.2.11 Setting the Tool Shape

Press the menu [Tool setting] to display the tool management screen. Set the tool shape used in the solid display for graphic check on the tool management screen.

Even though the menu [T-Mng.] is pressed on the [Setup] screen, the same screen can be displayed. For details, refer to "6.5 Setting the Tool Management Data (Tool Management Screen)".

4.2.12 Associating Graphic Check Images with a Machining Program

Refer to "4.1.14 Associating Graphic Check Images with a Machining Program".

4.2.13 Setting the Drawing Speed of Program Check

Check speed of graphic check can be accelerated. When accelerating, set the parameter "#8997 Hi-speed grph chck" to "1" to "3" (Level 1 to Level 3). However, the shape of workpiece drawing becomes the one taking inner route than the path commanded by the program.

When you want to confirm the shape with high-accuracy, disable the high-speed graphic check.

High-speed graphic check invalid	High-speed graphic check valid
The shape can be confirmed with high-accuracy. The check speed is low. 	The check can be performed at high-speed. Minute shape collapses. 

Note

(1) The setting of high-speed graphic check is not reflected while [Check continu] or [Check step] is operated in the background.

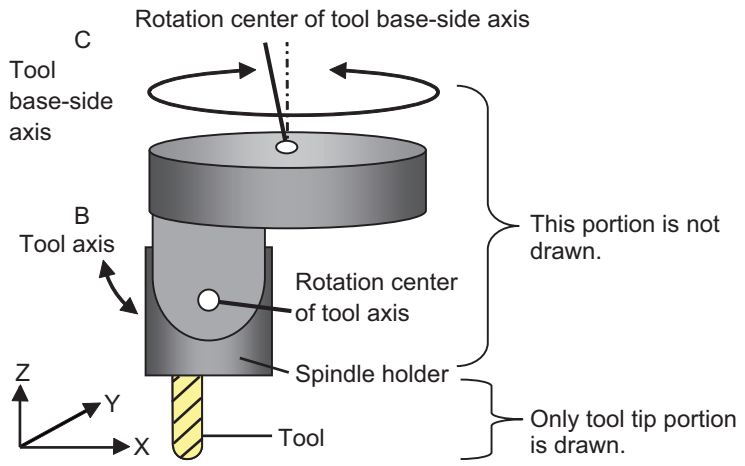
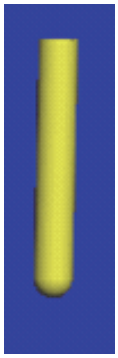
(2) High-speed graphic check is valid only for check method I.

4.2.14 Drawing Travel Command of the Rotary Axis

This chapter explains how to draw the travel command of the rotary axis.

Drawing travel command of the rotary axis on tool-side

Only tool tip is drawn.

Actual machine	Drawing in 3D check drawing area
	Tool other than the tool tip is not drawn. 

[Drawing travel command of tool base-side axis]

The travel command of the tool base-side axis is drawn based on the setting of the following rotary axis configuration parameters.

Parameter No.	Name	Details
#7920	SLCT_T1	Rotary axis selection
#7922	ROTAXT1	Rotary axis name
#7923	DIR_T1	Rotation direction
#7924	COFST1H	Horizontal axis rotation center offset
#7925	COFST1V	Vertical axis rotation center offset
#7926	COFST1T	Height axis rotation center offset

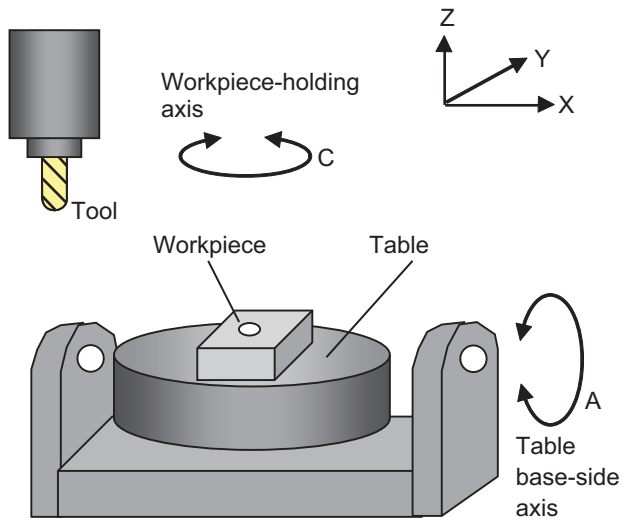
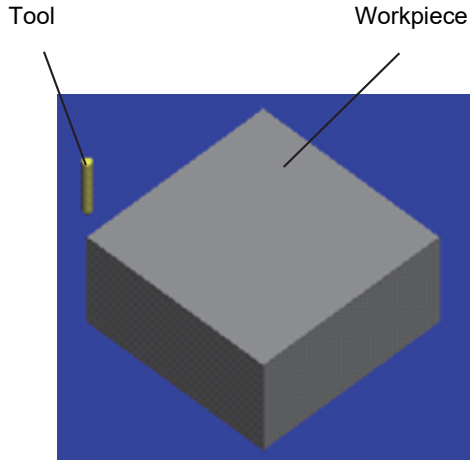
[Drawing travel command of tool axis]

The travel command of the tool axis is drawn based on the setting of the following rotary axis configuration parameters.

Parameter No.	Name	Details
#7930	SLCT_T2	Rotary axis selection
#7932	ROTAXT2	Rotary axis name
#7933	DIR_T2	Rotation direction
#7934	COFST2H	Horizontal axis rotation center offset
#7935	COFST2V	Vertical axis rotation center offset
#7936	COFST2T	Height axis rotation center offset

Drawing travel command of table-side rotary axis

The workpiece placed on the table is drawn, but the table itself is not drawn. Only one workpiece is drawn.

Actual machine operation	Drawing in 3D check drawing area
	 Table is not drawn.

[Drawing travel command of table base-side axis]

The travel command of the table base-side axis is drawn based on the setting of the following rotary axis configuration parameters.

Parameter No.	Name	Details
#7940	SLCT_W1	Rotary axis selection
#7942	ROTAXW1	Rotary axis name
#7943	DIR_W1	Rotation direction
#7944	COFSW1H	Horizontal axis rotation center offset
#7945	COFSW1V	Vertical axis rotation center offset
#7946	COFSW1T	Height axis rotation center offset

When the travel command is issued for the table base-axis, the tool moves while the workpiece appears to stand still.

[Drawing travel command of the workpiece-holding axis]

The travel command of the workpiece-holding axis is drawn based on the setting of the following rotary axis configuration parameters.

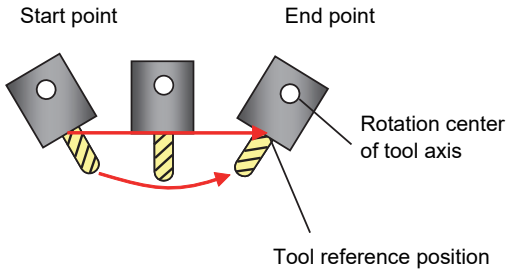
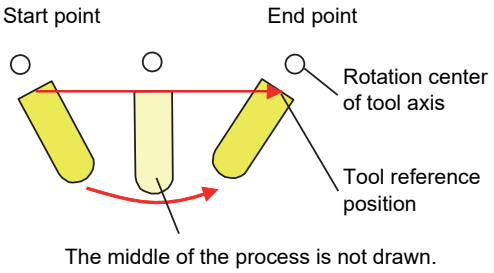
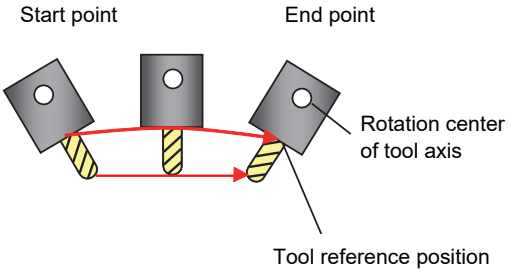
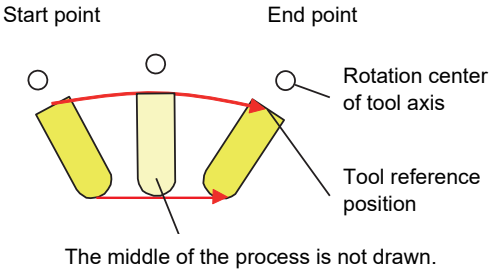
Parameter No.	Name	Details
#7950	SLCT_W2	Rotary axis selection
#7952	ROTAXW2	Rotary axis name
#7953	DIR_W2	Rotation direction
#7954	COFSW2H	Horizontal axis rotation center offset
#7955	COFSW2V	Vertical axis rotation center offset
#7956	COFSW2T	Height axis rotation center offset

When the travel command is issued for the workpiece-holding axis, the tool moves while the workpiece appears to stand still.

Drawing travel command during tool center point control/inclined surface machining command

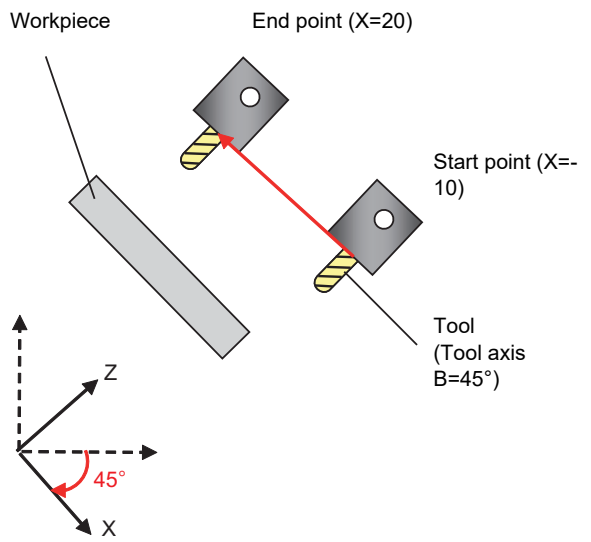
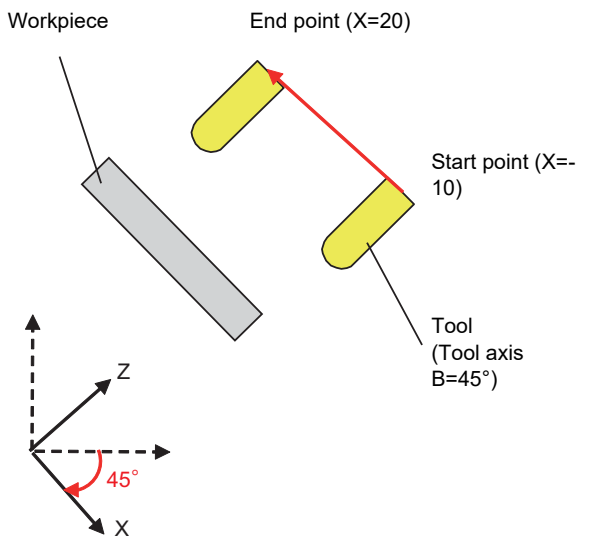
[Drawing travel command during tool center point control]

The drawing is as follows based on tool center point control valid (G43.4, G43.5) or tool center point control invalid (G49).

Tool center point control	Actual machine operation	Drawing of 3D check
Disabled (G49)	The position commanded with the machining program moves so that it becomes the tool base position. 	The position commanded with the machining program is drawn so that it becomes the tool base position. 
Enabled (G43.4, G43.5)	The position commanded with the machining program moves so that it becomes the center of tool tip on the coordinate system which rotates with the workpiece. 	The position commanded with the machining program is drawn so that it becomes the center of tool tip on the coordinate system which rotates with the workpiece. 

[Drawing travel command during inclined surface machining command]

The drawing during the inclined surface machining is as follows.

Actual machine operation	Drawing of 3D check
The axis moves on the inclined coordinate system (feature coordinate system) 	The axis moves on the inclined coordinate system (feature coordinate system) 

Drawing travel command without setting the rotary axis configuration parameters

When the rotary axis configuration parameters are not set, only the workpiece-holding axis is set as the rotary axis.
In this case, travel command of the workpiece-holding axis is drawn according to the setting of the following parameters.

Parameter No.	Name	Details
#1516	mill_ax	Milling axis name
#1533	millPax	Pole coordinate linear axis name
#1761	cfgPR11/bit0	Plane selection method for Polar coordinate interpolation
#12650	table center posH	Horizontal axis table center position
#12651	table center posV	Vertical axis table center position
#12652	table center posT	Height axis table center position

The settings of #1533 and #1761 define around which axis the rotary axis set in #1516 rotates.

4.2.15 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	A mirror image is applied and a check drawing is made at the setting status prior to starting the check.
Mirror image by external input	[Check method I (Foreground), Check method II] The signal is reflected, and check a drawing is carried out. [Check method I (Background)] The signal is ignored, and check a drawing is carried out.
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. A check drawing is performed with override fixed at 100% ignoring the signals (check method II).
Auto operation pause (feed hold)	Signals are ignored and a check drawing is made.
Auto operation start-up (cycle start)	If you press the cycle start button during checking, the operation alarm "M01 program check mode" is displayed (check method I (foreground) or check method II) while the button is held down. The operation can be started automatically even during check drawing (check method I (background)).
Auto-restart	The check is completed at M02/M30 and is not restarted.
Single block	A check drawing is performed ignoring the signals (check method II).
Manual arbitrary reverse run	A check drawing is performed ignoring the signals (check method II).
High-speed simple program check	A check drawing is performed ignoring the signals (check method II).
Tool life management	The tool life data count is not performed (check method II).
Skip (G31)	The skip signal is disabled. A drawing moves to the end point and is completed (check method II).
Workpiece coordinate offset	Offset is enabled (the target is drawn at the position determined with the designated offset amount).
Manual mode/handle	If the mode is switched to the manual mode during checking, the operation alarm "M01 program check mode" is displayed (check method I (foreground) or check method II). During checking, you can switch to the manual mode, HANDLE, to move the machine. However, the drawing position of the check is not affected by the mode change (check method I (background)).

Function name	Remarks
Tape operation and MDI operation	If the mode is switched to the manual mode during checking, the operation alarm "M01 program check mode" is displayed (check method II).
F1-digit Feed	The speed does not affect the check drawing with check method I.
User macro I, II	Basic variable calculations, decisions and branch functions are all enabled. However, the following functions are ignored. •Macro interface input/output •Single block stop •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 and 2nd miscellaneous function commands	M, S, T and 2nd miscellaneous function command signals commanded by the program during the check are not output. Neither do they display on the operation screen etc. The auxiliary function command by manual operation on the position display screen is output even during checking. With the setting of the parameter "#1284 ext20/bit3", not outputting/outputting the miscellaneous function can be switched (check method II).
G command modal	G modal displays on the modal display screen while the check drawing is being made. The modal information after the check cannot be returned to the state before the check (check method II).
Optional Block Skip	The setting status when starting the check drawing are used (check method I (foreground) or check method II). Use the menu to switch ON and OFF. The optional block skip state is held until the power is turned ON again, which is common to part systems. However, when the power is turned ON again, the mode is always turned OFF (check method I (background)).
Spindle position control (Spindle/C axis control)	Set the spindle position control (spindle/C axis control) for the part system to be checked to spindle mode, then execute a program check. Executing the check in C axis mode causes the operation error "M01 SP-C ax ctrl runs indepndntly 1026", and the check cannot be started. (Check method II) When the program command method is selected, the spindle/C axis moves in the C axis mode. (Check mehtod I)

Note

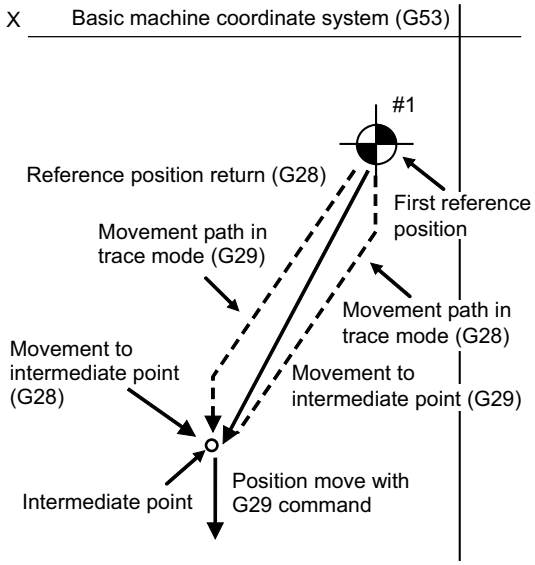
(1) Reference position return and start position return

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.



4.2.16 Handling of Variable Commands, Programmable Parameter Input, and Programmable Compensation Data Input

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. The setting commands, which are executed to various data during program check, are reflected on only drawing of background check. The actual machining operation is not affected.

Meaning of symbols in the table
○: The value returns to the status prior to starting the program check (Actual machining data is not rewritten).
×: The value does not return to the status prior to starting the program check (Actual machining data is rewritten).
Δ: The value varies depending on the parameter setting.

■Check method I (foreground)

Data type	Data status	Explanation
Parameters	×	The programmable parameter input commands which have been executed at the program check are set as actual data.
Local variables	Δ	It is possible to return to the status prior to starting the program check based on the value of the parameter "#1231 set03/bit0" (Graphic check compatibility parameter). 0: Returns to original status after check completion 1: Does not return to original status after check completion
Common variables	Δ	
Workpiece coordinate system offset	Δ	
Tool compensation amount	Δ	
Extended common variables	×	The extended common variable input commands which have been executed at the program check are set as actual data.

■Check method I (background)

Data type	Data status	Explanation
Parameters	○	The programmable parameter input commands which have been executed at the program check are not set as actual data.
Local variables	○	"#1231 set03/bit0" (Graphic check compatibility parameter) is disabled.
Common variables	○	
Workpiece coordinate system offset	○	
Tool compensation amount	○	
Extended common variables	○	

■Check method II

Data type	Data status	Explanation
Parameters	×	The programmable parameter input commands which have been executed at the program check are set as actual data.
Local variables	×	"#1231 set03/bit0" (Graphic check compatibility parameter) is disabled.
Common variables	×	
Workpiece coordinate system offset	×	
Tool compensation amount	×	
Extended common variables	×	

4.2.17 Notes for Graphic Check (3D)

(1) The check function is valid for only NC internal operation results. Thus, the path is not drawn for the command which requires machine movement.

(2) For the handling of various types of data being set in the program, refer to "4.2.16 Handling of Variable Commands, Programmable Parameter Input, and Programmable Compensation Data Input".

(3) "Program check completed" appears with M02/M30 command.

(4) Graphic check is not available for serial and MDI program.

(5) Check search is not available for the program whose file name is consist of 33 characters or more.

(6) When the check is executed, the drawing starts from the machine coordinate zero point. However, the machine coordinate zero point is (0, 0, 0) even when the value is set to the parameter "#2037 G53ofs" (zero point return).

(7) The drawing coordinate system settings of 2D and 3D graphic check are not linked.

(8) The origin mark is not drawn on the 3D drawing parts.

(9) If the screen is switched during checking, the check reset occurs and the check is interrupted.

(10) If tracing is turned ON during automatic operation, the menus such as [Check search], [Check continu], [Check step], and [Check reset] are grayed out. You cannot perform 3D solid check.

(11) The drawing is not initialized even if the performance such as enlarge, reduce, move, change display mode, or rotate is set during check drawing. The program does not stop, either.

(12) If the part system is switched during checking, the check is reset.

(13) The tool shape is drawn according to the parameter "#8920 3D tool ofs select" parameter and according to the compensation amount (radius and length) (corresponding to the T command number) and the values (tool radius, tool length, and shape data 1 and 2) being set on the 3D Tool Shape setting screen.
Therefore, if the tool length or compensation number being assigned during tool radius compensation differs from the T command number, a misalignment occurs between the tool shape and machine position, resulting in the 3D workpiece being machined excessively or not enough.

(14) When the tool change is specified and if the following data of the tool corresponding to the T number is zero, the tool is drawn as the default tool.

- If the tool type is not specified: Drill tool
- If the tool length is 0: 150-mm tool length
- If tool radius is 0: 20-mm tool radius
- If shape data 1 is 0: 1-mm shape data 1 (it is 5 mm for the tap of tool type)
- If shape data 2 is 0: 1-mm shape data 2 (Tool diameter x 0.05 mm for the tap of tool type)
- If the color is not designated: Red tool

(15) If the parameter "#8920 3D tool ofs select" is set to "0" or "1" and if the tool radius to be displayed is 0 or less, the 20-mm default tool radius is assumed and drawn.

(16) If the micro segment program or complex shape program is executed for 3D checking, the "Solid check disabled (memory shortage)" message is displayed and the drawing may be erased. In such case, divide the program into subprograms and repeat check drawing. Before starting check drawing, press the menu [Work init.] once.

(17) If the coordinate system is set to "Work Coord", the workpiece coordinate position is not changed even when a coordinate system switching command is included in the program that executes the program check. Therefore, the drawings may be overlapped and displayed.

(18) If the coordinate system is set to "Work Coord" and if a coordinate system switching command is not included in the program that executes the program check, the currently specified coordinate system is used and, therefore, drawing is not executed. Make sure the coordinate system is specified in the program.

(19) The "Tapped Color", that is the same as when the tapping tool is selected, is used on the "Work set" screen. Therefore, switching of the tap surface color and normal surface color depends on not G command but the selected tool. However, even with the tapping tool being selected, if it is moved in the arc or helical direction, the "Work set" screen does not have the "Tapped Color".

(20) During 3D check, the tap is drawn as a straight rod and the shape data is not reflected (refer to "4.2.11 Setting the Tool Shape").

(21) When there is a table per part system, drawing is performed as if one workpiece is placed on the table of the part system that started the check. Multiple workpieces cannot be drawn simultaneously.

(22) When the workpiece shape is changed from the external source such as another application, the drawing reflect to the workpiece shape on the 3D check screen by pressing the [Work setting] menu to display workpiece shape setting screen.

(23) If a travel command is issued during polar coordinate interpolation or circular interpolation when any of the following conditions is met, the message "Related parameters not set" appears and the graphic of the tool movement is not drawn.

- All of the rotary axis configuration parameters (#7900, #7901 and #7902) are not set, and #1516 is not set.
- Only tool-side rotary axis is set in the rotary axis configuration parameters.

(24) Set a different axis for each base axis parameter I, J or K (#1026 to #1028). If a same axis is set, the path might be drawn differently from the actual machining.

Notes for check method I (foreground)

(1) If you press the cycle start button when an arbitrary part system is being checked, the operation alarm "M01 program check mode" is generated and the cycle start is disabled while the button is held down.
Press the auto start button after the check is completed or after the check is stopped with the menu [Check reset].

(2) If you carry out the operation search or check operation (continuous or step check) during cycle start or automatic stop of an arbitrary part system, an operation message "Executing automatic operation" is displayed, and no search or check drawing is carried out.

(3) "Program check completed" appears with M02/M30 command. For the tape mode, the rewind process is not carried out.

(4) The check ends with the reset (NC reset or the menu [Check reset]).
When a program error occurs, cancel the error with the operation indicated above.
If NC reset is selected when the check screen is being displayed, check drawing ends but the menu to switch to the 2D check or 3D check is kept invalid. To enable this switching, press the menu [Check reset].

(5) During automatic operation, the contents of automatic operation program are displayed by the "Currently executed machining program" (ONB display) and "Buffer display" on the check screen. During checking, the contents of the checking program are displayed by the "Currently executed machining program" (ONB display) and "Buffer display" on the monitor screen.

(6) If you carry out the operation search or check operation (continuous or step check) when the program is restarted, the "Program restarting" operation message is displayed, and no search or check drawing is carried out.

(7) If the manual operation mode is selected during checking, the "M01 program check mode" is displayed and the axis cannot be moved. If you need to move the axis, temporarily stop the current check by executing a reset and then move the axis. When you switch from the manual operation mode to the memory mode, the alarm is released. You can continue the current check even if an alarm is output.

(8) When the PLC Interrupt signal is turned ON during checking, the "M01 Program check mode" is selected. To start the PLC interrupt, terminate the current check by executing a reset. When you turn the PLC interrupt signal OFF, the alarm is released. You can continue the current checking even if the PLC Interrupt signal is ON.

(9) On the maintenance screen, when the rotary axis configuration parameter is changed to display 3D check screen, the machining result of the workpiece is initialized.

(10) This does not support the dynamic switch of the rotary axis configuration parameter ("#1450 5axis_Spec/bit2" (application of rotary axis configuration parameters)) by inclined surface machining command. The parameter set in the own part system at the start of 3D check is valid.

(11) When the M01 is commanded with the menu [M0/M1 neglect] disabled, the drawing stops regardless of the state (ON or OFF) of the optional stop switch.

Notes for check method I (background)

(1) The program that you have selected during Check Search is not the program that is executed during automatic operation. Search the program for automatic operation by using the operation search function of the monitor screen.

(2) The parameters and compensation amount do not reflect on check drawing even if you set them during checking. To reflect them on the drawings, reset the check once. Then, press the menu [Check continu] or [Check step] and execute the check again.

(3) "Program check completed" appears with M02/M30 command.

(4) The check is not terminated via execution of a reset.
If a program error occurs, release the error by selecting the menu [Check reset].

(5) During check search, if the length of the full path name exceeds 48 characters, you cannot move to another directory.

(6) If the menu [Opt skip] is ON (highlighted), the blocks that have the "/" character are skipped and the T-code list is obtained.

(7) The file name continues to be displayed as an O-number display even if you erase or rename the file after it is checked and searched. If you check a file in this status, the "Push [Check reset] menu" message is displayed.

(8) During checking, if you read the integrated time by using variable number #3001 or #3002, the checking start time is obtained. You cannot write the integrated time.

(9) When [Check continu] is executed, even if variable number #3006 is used for checking, you cannot stop the program and you cannot display the message.

(10) When the M01 is commanded with the menu [M0/M1 neglect] disabled, the drawing stops regardless of the state (ON or OFF) of the optional stop switch.

Notes for check method II

(1) If you press the cycle start button when an arbitrary part system is being checked, the operation alarm "M01 program check mode" is generated and the cycle start is disabled while the button is held down.
Press the auto start button after the check is completed or after the check is stopped with the menu [Check reset].

(2) If you carry out the operation search or check operation (continuous or step check) during cycle start or automatic stop of an arbitrary part system, an operation message "Executing automatic operation" is displayed, and no search or check drawing is carried out.

(3) "Program check completed" appears with M02/M30 command. For the tape mode, the rewind process is not carried out.

(4) The check ends with the reset (NC reset or the menu [Check reset]).
When a program error occurs, cancel the error with the operation indicated above.
If NC reset is selected when the check screen is being displayed, check drawing ends but the menu to switch to the 2D check or 3D check is kept invalid. To enable this switching, press the menu [Check reset].

(5) During automatic operation, the contents of automatic operation program are displayed by the "Currently executed machining program" (ONB display) and "Buffer display" on the check screen. During checking, the contents of the checking program are displayed by the "Currently executed machining program" (ONB display) and "Buffer display" on the monitor screen.

(6) If you carry out the operation search or check operation (continuous or step check) when the program is restarted, the "Program restarting" operation message is displayed, and no search or check drawing is carried out.

(7) If the operation mode is switched to other than of memory mode during checking, the "M01 program check mode" is displayed and the check is stopped. When the mode returns to the memory mode again, the alarm is canceled and the check can be carried on.

(8) When the PLC Interrupt signal is turned ON during checking, the "M01 Program check mode" is selected. To start the PLC interrupt, terminate the current check by executing a reset. When you turn the PLC interrupt signal OFF, the alarm is released. You can continue the current checking even if the PLC Interrupt signal is ON.

(9) If the value is rewritten by G10L70 (parameter command by program), system variables or common variables in the program being checked, the value is rewritten in the same manner as when the program is executed during automatic operation.

(10) On the maintenance screen, when the rotary axis configuration parameter is changed to display 3D check screen, the machining result of the workpiece is initialized.

(11) When the sub part system control I/II commands are executed, the program error (P803) occurs and the check is stopped.

(12) During tap retract enable state, if the check operations ([Check continu] or [Check step]) is performed, "M01 Wait for tap retract 0057" is displayed and the check cannot performed in the part system in which the alarm occurred. After the state becomes tap retract enable and tap retract enable operation is completed, the check can be performed.

(13) When you use the arbitrary axis exchange control function, set all the part systems to be check targets for the graphic check (*1). When any axis in the part system which is not to be checked is designated, the operation error "M01 Arbitrary axis unexchangeable 1101" occurs, the graphic check cannot be performed in the part system where an alarm is displaying. Change the setting of the parameter or change the machining program.

(*1) The part system which is set to "0" by any of the parameters "#8982 CheckSmltns$1Invld" to "#8989 CheckSmltns$8Invld"

(14) It may take time to draw a helical interpolation path because it is drawn to the end point at a time.

Full-screen display of graphic check function (3D)

(1) For 3D check, the drawn workpiece shape is enlarged or reduced to be 90% of drawing area in accordance with the size of graphic drawing area when the screen is switched between normal display and full-screen display.

Setting for base axes

When a rotary axis is included among the base axis parameters I, J, and K (#1026 to #1028), set the base axis parameters for graphics I, J, and K (#8631 to #8633) to exclude the rotary axis as the following example.

If all the base axis parameters for graphics I, J, and K are not set, the graphic check operates in accordance with the setting of the base axis parameters #1026 to #1028.

[Setting example of parameters]

Base axis parameter

No.	Parameter name	Setting value	Explanation
#1026	base_I	X	Linear axis
#1027	base_J	C	Rotary axis
#1028	base_K	Z	Linear axis

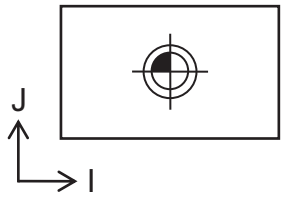
Base axis parameter for graphics

No.	Parameter name	Setting value	Explanation
#8631	GraphicsBaseAxis_I	X	Set the same axis name as #1026.
#8632	GraphicsBaseAxis_J		Enter "0" to the rotary axis to be blank.
#8633	GraphicsBaseAxis_K	Z	Set the same axis name as #1028.

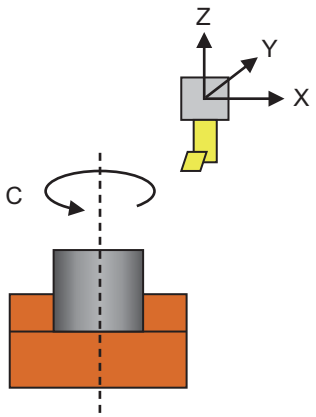
Using the lathe turning tool

Note the following points when the program check is performed using the lathe turning tool.

(1) Set the tool so that the coordinates I and J of the material zero point in the [Work setting] are in the center of the workpiece.
If the zero point is not in the center of the workpiece, the image of turning machining cannot be drawn correctly.



(2) For the graphic check using the lathe turning tool, C axis is configured as a spindle.

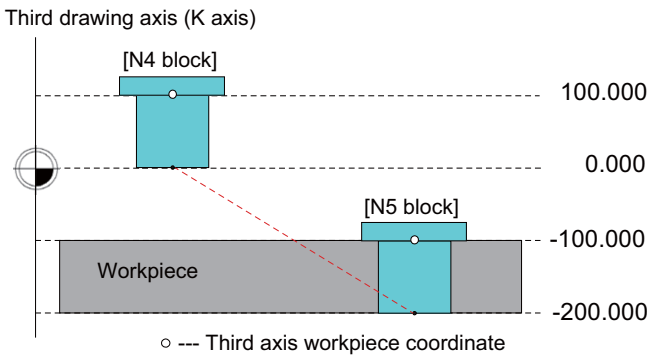


(3) For the lathe turning tool, the movement of the Y axis is not reflected in the drawing of the machining shape. The Y axis position is drawn as it is always positioned at workpiece zero point.

Drawing position

Set the drawing position with the parameter "#8920 3D tool ofs select" in accordance with the tool length measurement type (I/II).

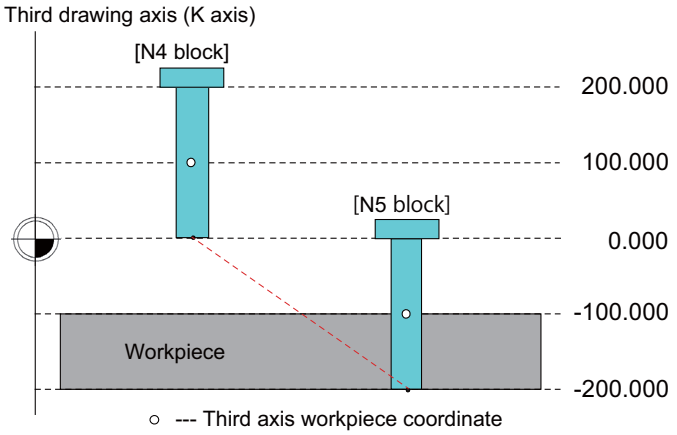
The following table shows the drawing position and the diameter and length of the tool to be displayed based on the combination with the tool length measurement type (I/II).

<table>
<tr><th colspan="2">Setting value of #8920</th><th>Drawing position (X, Y, Z)</th><th>Radius and length of tool to be displayed</th></tr>
<tr><td>0</td><td>(For tool length measurement type I)
For tool radius compensation, the tool compensation amount set in the tool compensation screen is used. For tool length, the data set in the 3D tool management screen is used.</td><td>X: Machine position
Y: Machine position
Z: Machine position</td><td>Radius: Tool radius compensation amount
Length: Tool length in the 3D tool management screen</td></tr>
<tr><td>1</td><td rowspan="2">(For tool length measurement type II)
For both tool radius and tool length, the tool compensation amount set in the tool compensation screen is used.</td><td>X: Machine position
Y: Machine position
Z: Varies depending on the conditions below.
♦During G43 (offset in positive (+) direction) modal
Machine position - (Tool length compensation amount + wear compensation amount)
♦During G44 (offset in negative (-) direction) modal
Machine position + (Tool length compensation amount + wear compensation amount)
♦G49 (Offset cancel)
Machine position</td><td>Radius: Tool radius compensation amount
Length: Tool length compensation amount + wear compensation amount</td></tr>
<tr><td colspan="2"><Example of the drawing position when #8920 = 1>
[Tool compensation amount screen]
Number 50
Length dimension: 110.000
Length wear: -10.000
Radius dimension: 60.000
Radius wear: -10.000
[Machining program]
N1 G91 G28 Z0
G91 G28 X0 Y0
N2 T50 M6
N3 G90 G54 G00 X100. Y100.
N4 G43 Z0. H50
N5 G91 G01 Z-200. F100.
:
[Radius and length of tool to be displayed]
Length = 100.000 (110.000 + (-10.000))
Radius = 50.000 (60.000 + (-10.000))

</td></tr>
</table>

<table>
<tr><th colspan="2">Setting value of #8920</th><th>Drawing position (X, Y, Z)</th><th>Radius and length of tool to be displayed</th></tr>
<tr><td>2</td><td>(For tool length measurement type I)
For both tool radius and tool length, the data set in the 3D tool management screen is used.</td><td>X: Machine position
Y: Machine position
Z: Machine position</td><td>Radius: Tool radius set in the 3D tool management screen
Length: Tool length set in the 3D tool management screen</td></tr>
<tr><td>3</td><td rowspan="2">(For tool length measurement type II)
For both tool radius and tool length, the data set in the 3D tool management screen is used.</td><td>X: Machine position
Y: Machine position
Z: Varies depending on the conditions below.
♦During G43 (offset in positive (+) direction) modal
Machine position - (Tool length compensation amount + wear compensation amount)
♦During G44 (offset in negative (-) direction) modal
Machine position + (Tool length compensation amount + wear compensation amount)
♦G49 (Offset cancel)
Machine position</td><td>Radius: Tool radius set in the 3D tool management screen
Length: Tool length set in the 3D tool management screen</td></tr>
<tr><td colspan="2"><Example of the drawing position when #8920 = 3>
[3D tool management screen]
Number 50
Tool length: 200.000
Tool radius: 20.000

[Tool compensation amount screen]
Number 50
Length dimension: 110.000
Length wear: -10.000
Radius dimension: 60.000
Radius wear: -10.000

[Radius and length of tool to be displayed]
Length = 200.000
Radius = 20.000

[Machining program]
N1 G91 G28 Z0
G91 G28 X0 Y0
N2 T50 M6
N3 G90 G54 G00 X100. Y100.
N4 G43 Z0. H50
N5 G91 G01 Z-200. F100.
:

</td></tr>
</table>

Note

(1) During the G43/G44 offset, drawing position compensation is applied only to the third axis (base axis K). A graphic is drawn at the machine position in the first and second axes (base axes L and J).

(2) If a graphic is drawn based on the workpiece coordinate system, "Machine position" in the above chart is replaced with "Workpiece coordinate position".

Drawing start position

When the check is executed, the drawing starts from the position set in the parameter "#8217 Check start point".

The drawing is not operated between the check end position of last time and the check start position of this time.

[Check method I]

If reference position return is commanded during the check, the drawing start position moves to the position set in the parameter #8217.

[Check method II]

If reference position return is commanded during the check, the drawing start position moves to the position set in the zero point return parameter ("#2037 G53ofs" to "#2040 #4_rfp").

Note

(1) With check method II, the drawing starts from the actual machine position when the check is executed.

4.3 Graphic Check (3D) (L System)



Graphic check (3D) is a function that allows you to check the machining program movement path only by drawing without automatic operations. Therefore, the graphics can be drawn at a high speed and high accuracy, and the machining program can be checked.
The turning machining and milling on the front and back sides of a workpiece can be drawn with 3D check of the graphic check function.

This function requires workpiece and tool shape settings.

Note

(1) When changing the screen to the edit screen, display depends on the previous checking type (2D/3D). If 2D graphic data appears, press the menu [3D check]. Note that when transferring to this screen during graphic trace, 3D solid program check is displayed.

[Terms]

The following descriptions are the meanings of the terms used in this chapter.

Foreground: Active state operated by the user such as automatic operation on the NC

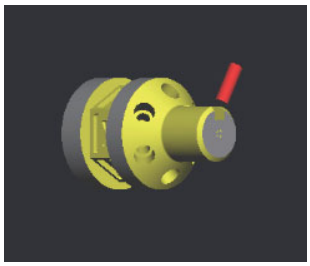
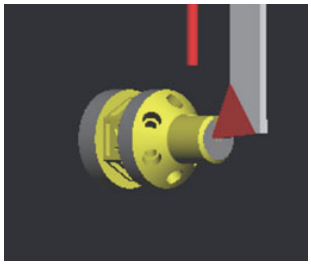
Background: Inactive state operated in the back of foreground

[Check method]

There are two types of 3D graphic check as below, which can be switched by [Type change] menu.

Note

(1) Make sure to select the check method before the check is started. During the check, [Type change] menu is grayed out, and the check method cannot be changed.

Check method	Outline of drawing	Details
Check method I	This method simplifies the machining result of machining program and simulates at a high speed. A machining program in single part system (displayed part system) is drawn. In 3D drawing area, one workpiece and one tool are displayed. 	This method is operated on the background on M800VW Series, M800VS Series and M80VW Series. This method is operated on the foreground on M80V Series.
Check method II	This method simulates the machining result of machining program at a higher accuracy than check method I. Machining programs in multiple part systems can be drawn simultaneously. In 3D drawing area, one workpiece and tools for part systems are displayed. 	This method is available only when the operation mode in the part system to be simulated is in the memory mode. This method is operated on the foreground and in the machine lock status during the program check. Also, during the program check, the PLC signal (Graphic check (Check method II) in progress X759) is turned ON. The program check cannot be started during automatic operation in arbitrary part system. The program check is in operation from the program execution by pressing the menu [Check continu] or [Check step] on the check screen to the following status. ♦Program ends. ♦The menu [Check reset] is pressed. ♦The reset signal or emergency stop signal outputs. ♦The screen is moved from the check screen to another one.

•Check method I

In the 2-part system or more, press the part system switch key $\$\leftrightarrow\$$ to switch the display to the target part system before starting checking. Note that program check is carried out in the 2-part system or more regardless of the command such as the timing synchronization between a hidden part system and another part system.

•Check method II

In two or more part systems, regardless of the displayed part system, the programs following the setting of the parameters "#8982 CheckSmltns$1Invld" to "#8989 CheckSmltns$8Invld" are checked simultaneously.

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

[Targets of drawing]

The workpiece and tool that can be drawn with the program check per part system are as shown below:

	Check method I	Check method II
Workpiece	1 workpiece Based on the premise that the workpiece is machined by the front-side spindle and the back-side spindle at a time.	
Tool	1 tool The tool used in the currently displayed part system is drawn. The part system can be switched.	Up to 8 tools The tool for the part system to be checked is drawn. Up to eight part systems can be drawn.

[Coordinate system during drawing]

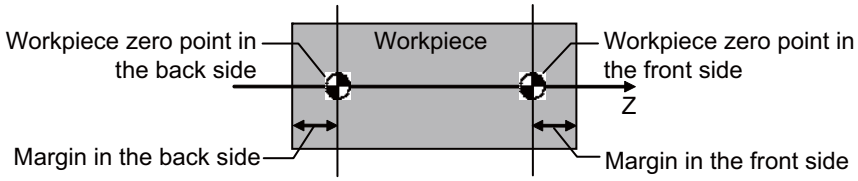
♦Check method I

3D graphic is drawn based on the programming coordinate system by setting the zero points in the following positions.

X-axis zero point: Center of workpiece

Y-axis zero point: Fix to 0

Z-axis zero point: Workpiece zero point Z (Zero point can be designated respectively for the front side and back side.)



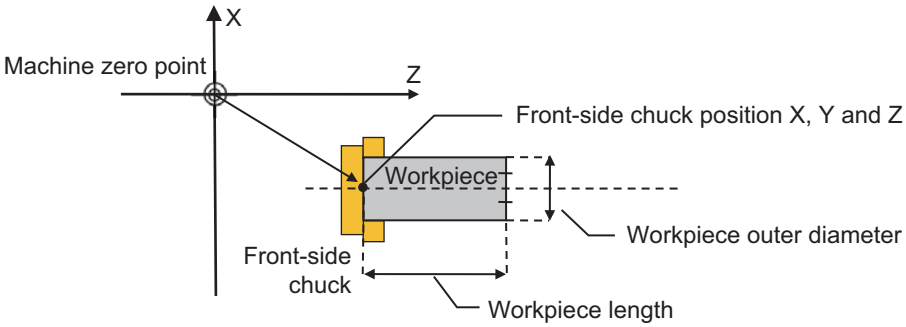
Refer to "4.3.10 Setting the Workpiece Shape" for details on setting the workpiece zero point Z.

♦Check method II

3D drawing is drawn in the machine coordinate system with the machine zero point as zero point. Therefore, the settings of front-side workpiece zero point Z and rear-side workpiece zero point set on the workpiece shape setting screen are treated as invalid.

The machine zero point is designated with the parameter "#2144 baseps" (Base machine coordinate zero point relative distance).

The display position of workpiece when the check is started.



For the chuck position, the setting by the MTB is required.

The chuck is not drawn in the 3D drawing area.

[Drawing shape of tool]

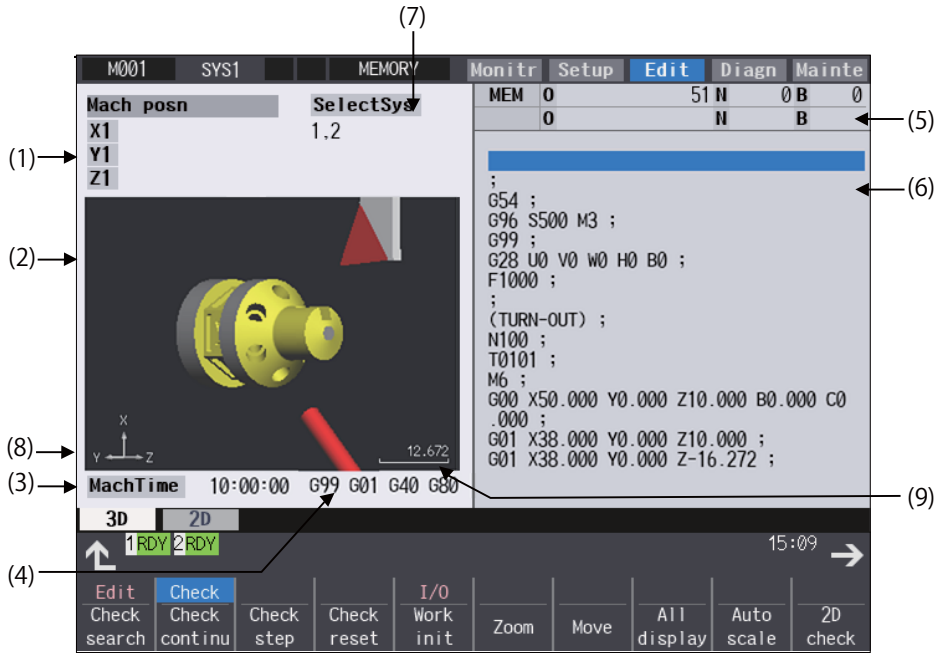
Turning tools are always drawn according to the tool information in the tool management screen; however, the shape of milling tools is set via the parameter (#8920 3D tool ofs select).

The shape of milling tools is set via the parameter (#8920 3D tool ofs select) as shown below:

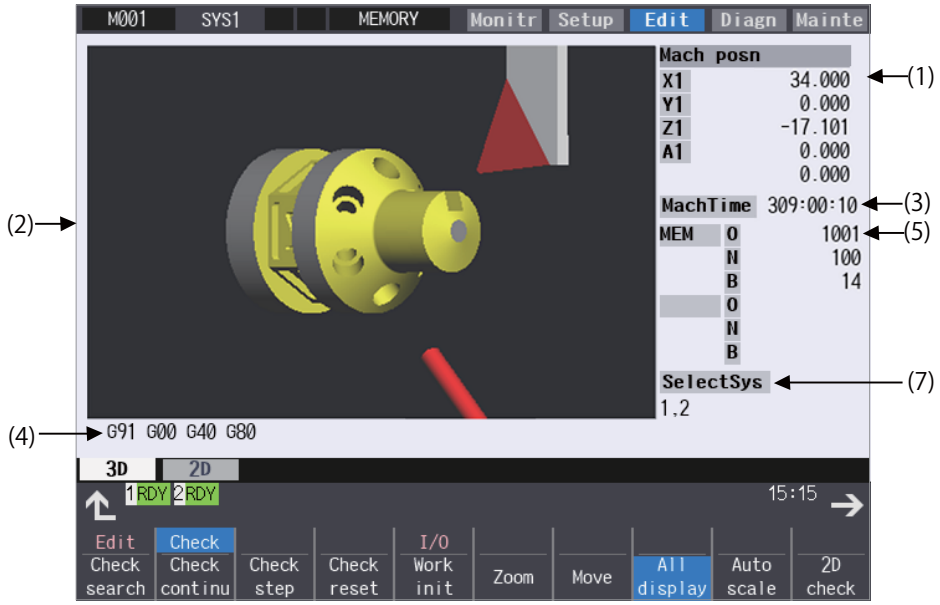
"3D tool ofs select" parameter		Radius and length of tool to be displayed
0	Tool radius: Tool compensation amount on the NC screen Tool length: Data on the tool management screen	Radius: Nose R compensation amount + wear compensation amount Length: Tool length "A" on the tool management screen
1	Tool radius: Tool compensation amount on the NC screen Tool length: Tool compensation amount on the NC screen	Radius: Nose R compensation amount + wear compensation amount Length: Tool length compensation amount + wear compensation amount
2	Tool radius: Data on the tool management screen Tool length: Data on the tool management screen	Radius: Tool length "B" on the tool management screen Length: Tool length "A" on the tool management screen

For the data on the tool management screen to be used for drawing tools, refer to "6.5 Setting the Tool Management Data (Tool Management Screen)".

■Normal display ([All display] is OFF)



■Full-screen display ([All display] is ON)



Display items

Display items			Details
(1)	Check counter		This displays the axis counter for the check drawing. In the normal display mode, the axis names common to part systems ("#1022 axname2") based on the basic axis name set by the parameters "#1026 base_I", "#1027 base_J", and "#1028 base_K" are displayed. In the full-screen display mode, maximum 5 axes are displayed. Three axes are set by #1026,#1027 and #1028, and two axes are set by the rotary axis configuration parameters. When the rotary axis configuration parameters are not set, rotary axes set by the parameter "#1013 axname" are sequentially displayed. [Check method I] This displays the workpiece coordinate counter for the current part system (the part system being checked). [Check method II] This displays the machine coordinate counter for the current part system (it is not necessarily the part system being checked).
(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.
(3)	Machining time display		The time required for machining is calculated and displayed.
(4)	G modal display for check		The following items are displayed from the left. ♦Feed per minute/Feed per revolution (G98, G99: Group 5) ♦Operation mode (G00, G01, etc.: Group 1) ♦Nose R compensation (G40, G41, G42: Group 7) ♦Fixed cycle (G70 etc.: Group 9) However, the next modal in M system may be displayed due to the switching command of program format. ♦Absolute/incremental (G90, G91: Group 3) ♦Operation mode (G00, G01, etc.: Group 1) ♦Tool compensation (G40, G41, G42: Group 7) ♦Fixed cycle (G70 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. When they exceed the displayable number of characters (12 characters), the excess is abbreviated and displayed (*).
		Sub O1234...	This displays the sub-program device name, program number, sequence number, and block number for the sub-program currently being executed. When they exceed the displayable number of characters (12 characters), the excess is abbreviated and displayed (*).
(6)	Buffer display		This displays the contents of the machining program currently being executed. The block being executed is highlighted. When the parameter "#19431 Num display in prg" is set to "1", the variable, operation or function in the numerical part that follows each address is replaced with the calculated value and displayed for the current block and the next block.
(7)	Part system to be checked (Check method II only)		This displays the part system to be checked (the part system for which the parameters "#8982 CheckSmltns$1Invld" to "#8989 CheckSmltns$8Invld" are set) (up to eight part systems). This menu is displayed with check method II only.
(8)	Coord display		This displays the coordinate system direction. This displays the axis direction of the base axis I/J/K with arrow.
(9)	Scale display		This shows the display range scale. The unit is set to "mm/80pixcel" for metric system and "inch/80pixcel" for inch system.

Menus

Menus	Details
Check search	This performs a check search. Select a program to be checked from the program file list displayed in the pop-up window.
Check continu	This checks the program continuously.
Check step	This checks the program per block.
Check reset	This resets 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 canceled.
Zoom	This enlarges/reduces the displayed workpiece shape.
Move	This moves the displayed workpiece shape.
All display	This switches the normal display mode and the full-screen display mode.
Auto scale	This enlarges/reduces the unmachined workpiece shape (stored in the NC) to be approximately 90% of the graphic area, and also initializes the rotation.
2D check	This changes the program check (2D) display. This menu cannot be selected while the check is performed or interrupted.
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.
Opt Skip	Select whether to execute a block that has a slash "/" in the program to be checked. Selecting "ON" skips a block with a slash "/" to carry out drawing. Selecting "OFF" also draws a block with a slash "/". The menu [Opt skip] works commonly between part systems. When this menu is set to ON (highlighted) in the 1st part system, it stays ON status even after the part system is switched, and when [Check continu] or [Check step] is performed, draw skipping the block with a slash "/".
Rotate	This rotates the displayed workpiece shape.
Regist. Image	This associates a graphic check image with a machining program and save the program.
Scale display	This displays the scale of coordinate system direction and display range in the graphic drawing area.
M0/M1 neglect	[Check method I] If [M0/M1 neglect] is ON, the program check does not stop even when M00/M01 is commanded. If [M0/M1 neglect] is OFF, the program check stops when M00/M01 is commanded. [Check method II] The menu [M0/M1 neglect] is grayed out and non-selectable. The program check does not stop in the block where M00/M01 is commanded. (*1)
Dsp Section	This displays the cross section of workpiece shape.
Type change	This switches between check method I and check method II. When check method switch is invalid (parameter "#1231 set03/bit7" is set to "1"), this menu is not displayed.

Menus	Details
Work setting	This sets the workpiece shape used at the solid display.
Tool setting	This sets the tool shape used at the solid display.

(*1) M code independent output (DM00/DM01) follows the parameter setting indicated below.
- #1284 ext20/bit3 (M code output during multi-system simultaneous simulation (Check type II))
- #1405 M_mode(SMLK) (M code output (during high-speed simple program check)
- #1451 M[M031-000](SMLK) to #1466 M[M511-480](SMLK) (Special operation registration M code (High-speed simple program check))

Note

(1) When automatic operation is running during the graphic trace mode, the menu [Check search], [Check continu], [Check step], [Check reset] cannot be selected and the 3D solid program check cannot be performed. If the program is not in the process of automatic operation, a check is possible only for 3D solid.

(2) The menus [2D check], [Check search], [Work setting] and [Tool setting] cannot be selected while the check is performed or interrupted.

(3) The menus [M0/M1 neglect] and [Regist. Image] are grayed out during continuous check, and the menus cannot be selected.

(4) The coordinate system direction and the scale display are held after the screen is switched, but they are not held after the power is turned OFF and ON again.

Sub-menus of [Check search]

Menus	Details
Memory	This selects the device for searching for the program. When a device with directory is selected, the root is selected first.
Memory2	
HD	
Memory card	
DS	
USB Memory	
Top jump	Display starts from the first page of the directory/list of files. The cursor moves to the top line.
Bottom jump	The last page of the directory/list of files is displayed. The cursor moves to the bottom line.
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.
Preview	This switches to show or hide the preview (detailed information).
Change Disp	This switches between list display mode and thumbnail display mode.
List update	This updates the list contents. (The latest contents of the currently selected device and directory are listed.)

Menus	Details
Sort change	This changes the method of sorting the list.
Close	This closes the pop-up window and quits this function.

4.3.1 Switching Check Method

Refer to "4.2.1 Switching Check Method".

4.3.2 Checking Continuously

Refer to "4.2.2 Checking Continuously".

4.3.3 Checking One Block at a Time

Refer to "4.2.3 Checking One Block at a Time".

4.3.4 Checking Multiple Part Systems Simultaneously

Refer to "4.2.4 Checking Multiple Part Systems Simultaneously".

4.3.5 Canceling the Program Check

Refer to "4.2.5 Canceling the Program Check".

4.3.6 Enlarging and Reducing the Workpiece Shape

Refer to "4.2.6 Enlarging and Reducing the Workpiece Shape".

4.3.7 Moving the Workpiece Shape

Refer to "4.2.7 Moving the Workpiece Shape".

4.3.8 Rotating the Workpiece Shape

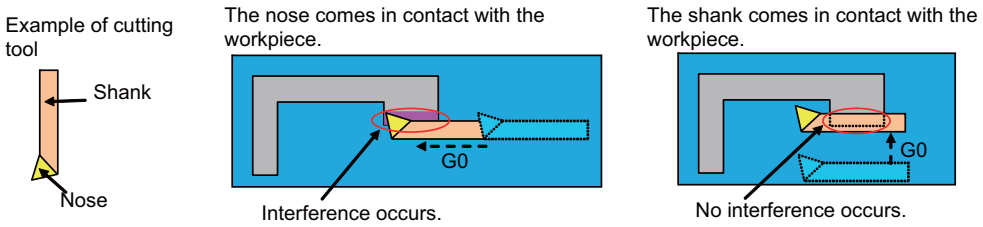
Refer to "4.2.8 Rotating the Workpiece Shape".

4.3.9 Performing an Interference Check

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.

It is possible to check whether the tool interferes with the workpiece during G0 movement before and after cutting.
The drawing does not stop even when interference occurs.
Because the shank part is fixedly drawn in a liner form, the shank part may interfere with a workpiece during inner diameter machining. Therefore, the tool nose part is the target of interference check but the shank part is not the target. (The part of workpiece where the shank interferes is not shaved.)

With the check method II, the interference is checked in all the part system to be checked. The interference between tools is not checked.



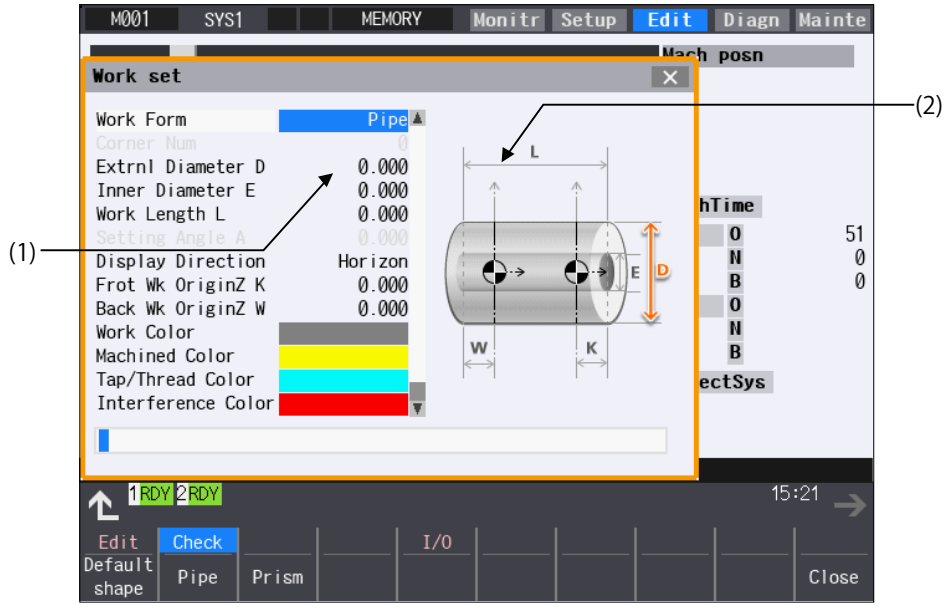
Operation method

(1) Press the menu [Colli check].

(2) Press the menu [Check step] or [Check continu]. ➡ The program check is executed for one block, and workpiece shape in the cutting process is drawn.
A "Checking" message is displayed and the menu [Check step] or [Check continu] is highlighted.
The workpiece and tool contact, this is drawn in a color highlighting the interference.

4.3.10 Setting the Workpiece Shape

Press the menu [Work setting] to display a pop-up window similar to the following, and set the workpiece shape used in the solid display.



Display items

Display items		Description
(1)	Workpiece shape setting area	This sets all workpiece shape items.
(2)	Workpiece setting 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 "Work Form" (material shape))

Menus	Description
Default shape	This sets the material shape. If a default shape is selected, it is drawn as a round bar. Designate [Pipe] for a round bar. Designate [Prism] for a square bar.
Pipe	
Prism	
Close	This closes the pop-up window and quits this function.

Menus (the cursor is at the viewing direction)

Menus	Description
Horiz	This designates the viewing direction of the workpiece.
Verti	

Menus (the cursor is at the "Work Color", "Machined Color", "Tapped Color", "Interference Color", or "Section Color")

Menus	Details
Default color Gray Red Yellow Blue Green Light blue Purple Pink	This sets the material color (Work Color), machining surface color (Machined Color), tapped/threaded surface color (Tapped/Threaded color), interference surface color (Interference Color) and section color (Section Color). 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 "Tapped Color", that is the same as when the tapping tool is selected, is used on the "Work set" screen. Therefore, switching of the tap surface color and normal surface color depends on not G command but the selected tool. However, even with the tapping tool being selected, if it is moved in the arc or helical direction, the "Work set" screen does not have the "Tapped Color".
Close	This closes the pop-up window and quits this function.

Menu (the cursor is at other item)

Menus	Details
Close	This closes the pop-up window and quits this function.

Setting the material shape (Work Form)

(1) Use the [↑] or [↓] key to move the cursor to the material shape settings column.

(2) Press the menu [Pipe].
It is also possible to set input values.
0 [INPUT]: Default shape
1 [INPUT]: Pipe
2 [INPUT]: Prism

→ "Pipe" appears at the cursor position.
The cursor moves to the next line.
A pipe guide drawing appears in the guide drawing area.

Setting the workpiece zero point Z

For the workpiece zero point, set the margin from the face of the material. For the margin, set the distance in the Z axis direction between the face and the workpiece zero point.

(1) Use the [↑] or [↓] key to move the cursor to the setting column of the front-side workpiece zero point Z.

(2) Input data.
(Example) 10 [INPUT]

→ "10.000" appears at the cursor position.
The cursor moves to the next line.

(3) Set the back-side workpiece zero point Z in the same manner.
If there is no back-side spindle, it is not necessary to set the back-side workpiece zero point Z.

Note

(1) The following table shows the setting range of the workpiece zero point.

(2) If a value outside of the setting range is entered, an error occurs, and the value cannot be set.

#1003 Input setup unit	#1041 Initial 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

(3) With the check method II, as the drawing is performed with the machine zero point as zero point, the values of this setting item are not used.

Setting the material outer diameter

Set the outer diameter of the material in millimeters in the millimeter system, and in inches in the inch system.

(1) Use the [↑] or [↓] key to move the cursor to the setting column of material outer diameter.

(2) Input data.
(Example) 100 [INPUT]
→ "100.000" appears at the cursor position. The cursor moves to the next line.

(3) Set other dimensional data items in the same manner.

Note

(1) The following table shows the setting ranges of the outer diameter of the material and other dimensional data items.

(2) If a value outside of the setting range is entered, an error occurs, and the value cannot be set.

#1003 Input setup unit	#1041 Initial inch	
	0 (mm)	1 (inch)
B	0.001 to 99999.999	0.0001 to 9999.9999
C	0.0001 to 99999.9999	0.00001 to 9999.99999
D	0.00001 to 99999.99999	0.000001 to 9999.999999
E	0.000001 to 99999.999999	0.0000001 to 9999.9999999

Setting the material color (WorkColor)

(1) Use the [↑] or [↓] key to move the cursor to the material color setting column.

(2) Press the material color menu.
(Example) [Gray]
→ The cell where the cursor is located appears in gray. The cursor moves to the next line. (The cursor does not move when at the cross section color.)

It is also possible to set input values.
0 [INPUT]: Default color
1 [INPUT]: Gray
2 [INPUT]: Red
3 [INPUT]: Yellow
4 [INPUT]: Blue
5 [INPUT]: Green
6 [INPUT]: Light blue
7 [INPUT]: Purple
8 [INPUT]: Pink

(3) Set the other color data 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/screw machining surface color: Light blue
  Interference surface color: Red
  Cross section color: Green

4.3.11 Setting the Tool Shape

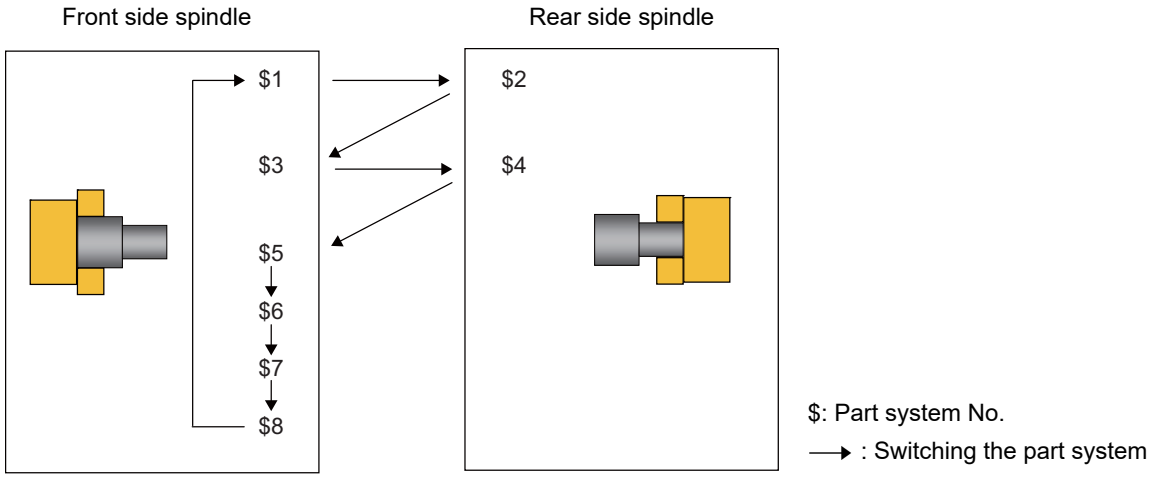
Press the menu [Tool setting] to display the tool management screen. Set the tool shape used in the solid display for graphic check on the tool management screen.

Even though the menu [T-Mng.] is pressed on the [Setup] screen, the same screen can be displayed. For details, refer to "6.5 Setting the Tool Management Data (Tool Management Screen)".

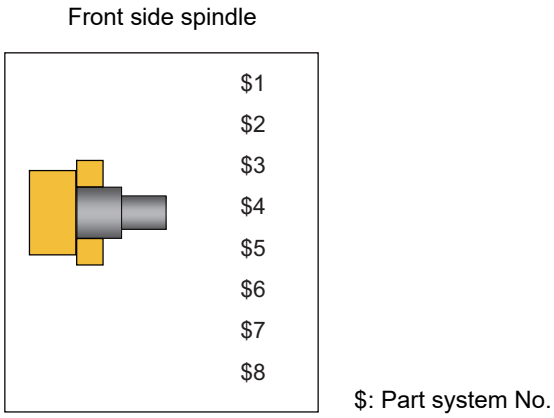
4.3.12 Checking a Machining Program between Part Systems

When checking multi-part system programs with the check method I, check a program of one part system first, and then switch the part system to check the other program. Multiple programs cannot be checked at a time with the check method I.

It depends on the part system as to whether a workpiece is mounted to the front side or rear side spindle when the check starts, as shown in the figure below.
When the rear side spindle exists (the parameter "#12628 sub sp no" is set to nonzero), and you switch the part system to 2nd or 4th, the workpiece is assumed to be mounted to the rear side spindle. When the rear side spindle exists, and switch the part system to other than 2nd and 4th, the workpiece is assumed to be mounted to front side spindle.



When the rear side spindle exists, and operation is started with the 2nd or 4th part system, the workpiece is assumed to be mounted to the rear side spindle.
When the rear side spindle exists, and operation is started with a part system other than 2nd and 4th, the workpiece is assumed to be mounted to the front side spindle.
When the rear side spindle does not exist, the workpiece is assumed to be mounted to the front side spindle.



When the spindle is switched between the front and rear sides, the workpiece zero point position Z will also be switched. Thus, the workpiece on the screen is not changed, but the coordinate of the tool will be displayed based on the workpiece zero point position Z of the spindle side.

If it is necessary to change the zero point position Z of the front or rear workpiece after switching the part system, change it on the "Work set " screen. When the parameter "#1231 set03/bit6" (Initialization of drawing in graphic check) is valid, the workpiece shape is retained if you change the setting values.

The setting items for a workpiece shape are common to all part systems. Thus when the part system is switched, the settings do not change. When executing the multi-part system program, and if you change any setting including the workpiece zero point position after switching between the front and rear side spindles, the workpiece is initialized. Therefore, when the parameter "#1231 set03/bit6" (Initialization of drawing in graphic check) is valid, the workpiece will not be initialized even if specific setting item for the workpiece shape is changed.

Refer to the following table to see whether the workpiece is initialized or not by changing each item for workpiece shape.

No.	Setting item for workpiece shape	Parameter "#1231 set03/bit6"	
		Enabled	Disabled
1	Work form	×	×
2	Number of corner	×	×
3	Workpiece outer diameter	×	×
4	Workpiece inner diameter	×	×
5	Workpiece length	×	×
6	Setting angle	×	×
7	Display direction	○	×
8	Front workpiece zero point position Z	○	×
9	Rear workpiece zero point position Z	○	×
10	Workpiece color	○	×
11	Machined color	○	×
12	Tapped/Threaded color	○	×
13	Interference Color	○	×
14	Section Color	○	×

×: Initialized ○: Not initialized

4.3.13 Associating Graphic Check Images with a Machining Program

Refer to "4.1.14 Associating Graphic Check Images with a Machining Program".

4.3.14 Setting the Drawing Speed of Program Check

Refer to "4.2.13 Setting the Drawing Speed of Program Check".

4.3.15 Availability of Check Mode with Other Functions

Refer to "4.2.15 Availability of Check Mode with Other Functions".

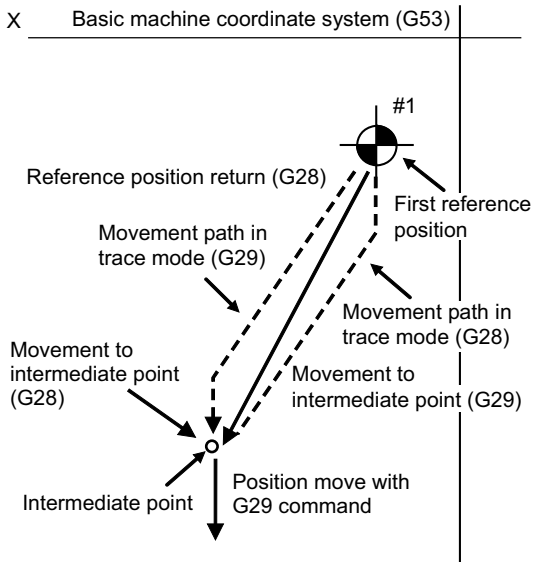
However, in the program check (3D) for L system, the relationship between the workpiece coordinate system offset and program format switch is as follows.

<table>
<tr><th>Function name</th><th>Remarks</th></tr>
<tr><td>Workpiece coordinate system offset</td><td>[Check method I]
The offset is disabled. A drawing is performed with the following position as zero point.
♦X-axis zero point: Center of workpiece
♦Z-axis zero point: Front-side workpiece zero point Z, rear-side workpiece zero point (set on the workpiece setting screen)
♦Y-axis zero point: Center of workpiece
♦Rotary axis zero point: 0 °
[Check method II]
The offset is enabled.</td></tr>
<tr><td>Program format switch</td><td>If M system G code shown below is commanded during program format switch, the machining result is different from the actual machining.

M system G code for which the machining result is different from the actual machining with check method I/II
<table>
<tr><th>G code</th><th>Function</th></tr>
<tr><td>G2.3/G3.3</td><td>Exponential interpolation</td></tr>
<tr><td>G41.2/G42.2</td><td>3-dimensional tool radius compensation</td></tr>
<tr><td>G43.4/G43.5</td><td>Tool center point control</td></tr>
<tr><td>G53.1/G53.6</td><td>Tool axis direction control</td></tr>
<tr><td>G61.4</td><td>Spline interpolation 2</td></tr>
<tr><td>G68.3</td><td>Inclined surface machining command (Designation with the tool axis direction)</td></tr>
<tr><td>G84</td><td>Fixed cycle (tapping)</td></tr>
</table>
M system G code for which the machining result is different from the actual machining with check method II
<table>
<tr><th>G code</th><th>Function</th></tr>
<tr><td>G7.1</td><td>Cylindrical interpolation</td></tr>
</table>
M system G code for which the machining result is different from the actual machining with check method I
<table>
<tr><th>G code</th><th>Function</th></tr>
<tr><td>G38</td><td>Tool radius compensation vector designation</td></tr>
<tr><td>G41/G42</td><td>Tool radius compensation</td></tr>
</table></td></tr>
</table>

Note

(1) 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.
With check method I, 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.



4.3.16 Handling of Variable Commands, Programmable Parameter Input, and Programmable Compensation Data Input

Refer to "4.2.16 Handling of Variable Commands, Programmable Parameter Input, and Programmable Compensation Data Input".

4.3.17 Notes for Graphic Check (3D)

(1) The check function is valid for only NC internal operation results. Thus, the path is not drawn for the command which requires machine movement.
(2) For the handling of various types of data being set in the program, refer to "4.2.16 Handling of Variable Commands, Programmable Parameter Input, and Programmable Compensation Data Input".
(3) "Program check completed" appears with M02/M30 command.
(4) Graphic check is not available for serial and MDI program.
(5) Check search is not available for the program whose file name is consist of 33 characters or more.
(6) When the check is executed, the drawing starts from the machine coordinate zero point. However, the machine coordinate zero point is (0, 0, 0) even when the value is set to the parameter "#2037 G53ofs" (zero point return).
(7) The drawing coordinate system settings of 2D and 3D graphic check are not linked.
(8) The origin mark is not drawn on the 3D drawing parts.
(9) If the screen is switched during checking, the check reset occurs and the check is interrupted.
(10) If tracing is turned ON during automatic operation, the menus such as [Check search], [Check continu], [Check step], and [Check reset] are grayed out. You cannot perform 3D solid check.
(11) The drawing is not initialized even if the performance such as enlarge, reduce, move, change display mode, or rotate is set during check drawing. The program does not stop, either.
(12) With the check method I, if the part system is switched during checking, the check is reset. With the check method II, the check is not reset.
(13) Tool shape is drawn using the value (shape data A to H) set in the tool management screen, regardless of the value of the parameter "#8920 3D tool ofs select".

(14) When the tool change is specified and if the following data of the tool corresponding to the T number is zero, the tool is drawn as the default tool.

Data	Value
Tool type	Lathe turning
Hand	Front/Right hand: except 2nd/4th part system
	Back/Right hand: 2nd/4th part system (setting of #1268 is nonzero.)
	Front/Right hand: 2nd/4th part system (setting of #1268 is "0".)
Tool length A	150 mm
Tool length B	25 mm
Nose angle	80°
Cutting edge angle	95°
Nose R	1 mm
Tool width	27.5 mm
GangTool ofs Y	0 mm

The tool to be displayed is switched to the tool corresponding to the T number when the tool exchange command is issued.

The previous tool is maintained and displayed until the next tool exchange command is issued.

It is not maintained when the power is turned on again. The default tool is displayed.

(15) When the tool change is specified and if the following data of the tool corresponding to the T number is zero, the tool is drawn as the default tool.

Turning tool

Turning		Groove		Thread cutting		Turning drill		Turning tapping	
Tool length A	150 mm	Tool length A	150 mm	Tool length A	150 mm	Tool length A	150 mm	Tool length A	150 mm
Tool length B	25 mm	Tool length B	25 mm	Tool length B	25 mm	Tool radius B	20 mm	Tool radius B	20 mm
Nose angle	80°	Nose R	1 mm	Tool width	10 mm	Nose angle	118°	Pitch	5 mm
Cutting edge angle	95°	Nose width	3 mm	GangTool ofs Y	0 mm	GangTool ofs Y	0 mm	Root dia.	1.25 mm
Nose R	1 mm	Maximum groove depth	12 mm	Color designation	Red	Color designation	Red	GangTool ofs Y	0 mm
Tool width	27.5 mm	Tool width	10 mm					Color designation	Red
GangTool ofs Y	0 mm	GangTool ofs Y	0 mm						
Color designation	Red	Color designation	Red						

Milling tool

- If the tool length is 0: 150-mm tool length
- If tool radius is 0: 20-mm tool radius
- If shape data 1 is 0: 1-mm shape data 1 (it is 5 mm for the tap of tool type)
- If shape data 2 is 0: 1-mm shape data 2 (Tool diameter x 0.05 mm for the tap of tool type)
- Gang tool offset Y is 0 mm.
- If the color is not designated: Red tool

(16) If the parameter "#8920 3D tool ofs select" is set to "0" or "1" and if the tool radius to be displayed is 0 or less, the 20-mm default tool radius is assumed and drawn.

(17) If the micro segment program or complex shape program is executed for 3D checking, the "Solid check disabled (memory shortage)" message is displayed and the drawing may be erased. In such case, divide the program into subprograms and repeat check drawing. Before starting check drawing, press the menu [Work init] once.

(18) With the check method I, the workpiece zero point is not changed even when a coordinate system switching command is included in the program that executes the program check. Therefore, the drawings may be overlapped and displayed.

(19) The "Tapped Color", that is the same as when the tapping tool is selected, is used on the "Work set" screen. Therefore, switching of the tap surface color and normal surface color depends on not G command but the selected tool. However, even with the tapping tool being selected, if it is moved in the arc or helical direction, the "Work set" screen does not have the "Tapped Color".

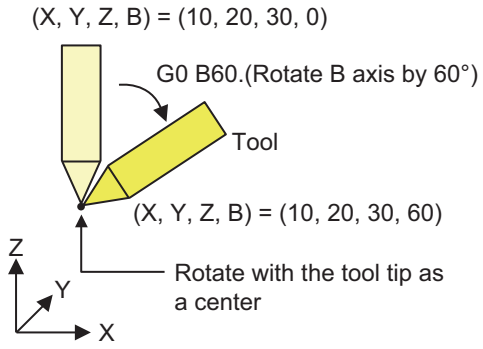
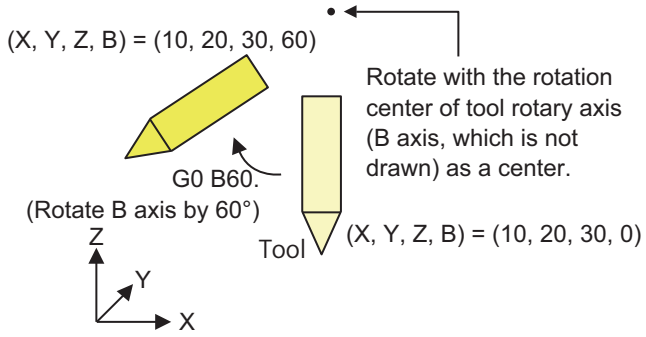
(20) During 3D check, the tap is drawn as a straight rod and the shape data is not reflected (refer to "4.3.11 Setting the Tool Shape").

(21) The check search cannot be executed when the sub part system that cannot be operated is displayed. If the search is carried out, the message "Search error" appears.

(22) If there is a phase difference between the front and rear side spindles when passing a workpiece, the graphic may not be drawn correctly.

(23) When the simple inclined surface machining is executed with the check method I, the counter on the check screen is displayed with the feature coordinate system.

(24) For the rotation of tool rotary axis, the rotation method differs depending on the check method, I or II (see the figure below).

Check method I	Check method II
Rotate with the tool tip as the center 	Rotate with rotation center of tool axis set with the rotary axis configuration parameters (#7930 to #7939) as the center 

(25) If such circular interpolation is executed that causes the angle between the start point and the end point to be 180 degrees or more during turning interpolation, the message "Draw library inside error(4)" appears. The tool moves to the end point of the circular interpolation; however, the workpiece is not machined.

(26) If a travel command is executed during polar coordinate interpolation or cylindrical interpolation on the machine which cannot perform C axis control on its workpiece spindle, the message "Draw library inside error(4)" appears and the tool does not move.

(27) If polar coordinate interpolation or circular interpolation is executed on the machine equipped with Y axis while Y axis is not at 0, the path is drawn differently from the actual machining.

(28) If polar coordinate interpolation or circular interpolation is executed while the axis for which [GangTool ofs Y] is set to nonzero is in use, the path is drawn differently from the actual machining.

(29) When the workpiece shape is changed from the external source such as another application, the drawing reflect to the workpiece shape on the 3D check screen by pressing the [Work setting] menu to display workpiece shape setting screen.

(30) Set a different axis for each base axis parameter I, J or K (#1026 to #1028). If a same axis is set, the path might be drawn differently from the actual machining.

(31) The operation of the eccentric turning command is replaced with the liner interpolation and drawn. The positions of the start and end points for the eccentric turning command block can be checked; however, the shape of the eccentric workpiece cannot be checked.

Notes for check method I (foreground)

(1) If you press the cycle start button when an arbitrary part system is being checked, the operation alarm "M01 program check mode" is generated and the cycle start is disabled while the button is held down.
Press the auto start button after the check is completed or after the check is stopped with the menu [Check reset].

(2) If you carry out the operation search or check operation (continuous or step check) during cycle start or automatic stop of an arbitrary part system, an operation message "Executing automatic operation" is displayed, and no search or check drawing is carried out.

(3) "Program check completed" appears with M02/M30 command. For the tape mode, the rewind process is not carried out.

(4) The check ends with the reset (NC reset or the menu [Check reset]).
When a program error occurs, cancel the error with the operation indicated above.
If NC reset is selected when the check screen is being displayed, check drawing ends but the menu to switch to the 2D Check or 3D Check is kept invalid. To enable this switching, press the menu [Check reset].

(5) During automatic operation, the contents of automatic operation program are displayed by the "Currently executed machining program" (ONB display) and "Buffer display" on the check screen. During checking, the contents of the checking program are displayed by the "Currently executed machining program" (ONB display) and "Buffer display" on the monitor screen.

(6) If you carry out the operation search or check operation (continuous or step check) when the program is restarted, the "Program restarting" operation message is displayed, and no search or check drawing is carried out.

(7) If the manual operation mode is selected during checking, the "M01 program check mode" is displayed and the axis cannot be moved. If you need to move the axis, temporarily stop the current check by executing a reset and then move the axis. When you switch from the manual operation mode to the memory mode, the alarm is released. You can continue the current check even if an alarm is output.

(8) When the PLC Interrupt signal is turned ON during checking, the "M01 Program check mode" is selected. To start the PLC interrupt, terminate the current check by executing a reset. When you turn the PLC interrupt signal OFF, the alarm is released. You can continue the current checking even if the PLC Interrupt signal is ON.

(9) When the M01 is commanded with the menu [M0/M1 neglect] disabled, the drawing stops regardless of the state (ON or OFF) of the optional stop switch.

Notes for check method I (background)

(1) The program that you have selected during Check Search is not the program that is executed during automatic operation. Search the program for automatic operation by using the operation search function of the monitor screen.

(2) The parameters and compensation amount do not reflect on check drawing even if you set them during checking. To reflect them on the drawings, reset the check once. Then, press the menu [Check continu] or [Check step] and execute the check again.

(3) "Program check completed" appears with M02/M30 command.

(4) The check is not terminated via execution of a reset.
If a program error occurs, release the error by selecting the menu [Check reset].

(5) During check search, if the length of the full path name exceeds 48 characters, you cannot move to another directory.

(6) If the menu [Optional Skip] is ON (highlighted), the blocks that have the "/" character are skipped and the T-code list is obtained.

(7) The file name continues to be displayed as an O-number display even if you erase or rename the file after it is checked and searched. If you check a file in this status, the "Push [Check reset] menu" message is displayed.

(8) During checking, if you read the integrated time by using variable number #3001 or #3002, the checking start time is obtained. You cannot write the integrated time.

(9) When [Check continu] is executed, even if variable number #3006 is used for checking, you cannot stop the program and you cannot display the message.

(10) When the M01 is commanded with the menu [M0/M1 neglect] disabled, the drawing stops regardless of the state (ON or OFF) of the optional stop switch.

(11) Even if the parameter "#19431 Num display in prg" is set to "1", the variable, operation or function is not replaced with the calculated value.

Full-screen display of graphic check function (3D)

(1) For 3D check, the drawn workpiece shape is enlarged or reduced to be 90% of drawing area in accordance with the size of graphic drawing area when the screen is switched between normal display and full-screen display.

Setting for base axes

When a rotary axis is included among the base axis parameters I, J, and K (#1026 to #1028), set the base axis parameters for graphics I, J, and K (#8631 to #8633) to exclude the rotary axis as the following example.

If all the base axis parameters for graphics I, J, and K are not set, the graphic check operates in accordance with the setting of the base axis parameters #1026 to #1028.

[Setting example of parameters]

Base axis parameter

No.	Parameter name	Setting value	Explanation
#1026	base_I	X	Liner axis
#1027	base_J	C	Rotary axis
#1028	base_K	Z	Liner axis

Base axis parameter for graphics

No.	Parameter name	Setting value	Explanation
#8631	GraphicsBaseAxis_I	X	Set the same axis name as #1026.
#8632	GraphicsBaseAxis_J		Enter "0" to the rotary axis to be blank.
#8633	GraphicsBaseAxis_K	Z	Set the same axis name as #1028.

Notes for check method II

(1) If you press the cycle start button when an arbitrary part system is being checked, the operation alarm "M01 program check mode" is generated and the cycle start is disabled while the button is held down.

(2) If you carry out the operation search or check operation (continuous or step check) during cycle start or automatic stop of an arbitrary part system, an operation message "Executing automatic operation" is displayed, and no search or check drawing is carried out.

(3) "Program check completed" appears with M02/M30 command. For the tape mode, the rewind process is not carried out.

(4) The check ends with the reset (NC reset or the menu [Check reset]).
When a program error occurs, cancel the error with the operation indicated above.
If NC reset is selected when the check screen is being displayed, check drawing ends but the menu to switch to the 2D Check or 3D Check is kept invalid. To enable this switching, press the menu [Check reset].

(5) During automatic operation, the contents of automatic operation program are displayed by the "Currently executed machining program" (ONB display) and "Buffer display" on the check screen. During checking, the contents of the checking program are displayed by the "Currently executed machining program" (ONB display) and "Buffer display" on the monitor screen.

(6) If you carry out the operation search or check operation (continuous or step check) when the program is restarted, the "Program restarting" operation message is displayed, and no search or check drawing is carried out.

(7) If the manual operation mode is selected during checking, the "M01 program check mode" is displayed and the check is stopped. If the mode returns to the memory mode again, the alarm is canceled and the check can be carried on.

(8) When the PLC Interrupt signal (YC2E) is turned ON during checking, the "M01 Program check mode" is selected. To start the PLC interrupt, terminate the current check by executing a reset. When you turn the PLC interrupt signal OFF, the alarm is released. You can continue the current checking even if the PLC Interrupt signal is ON.

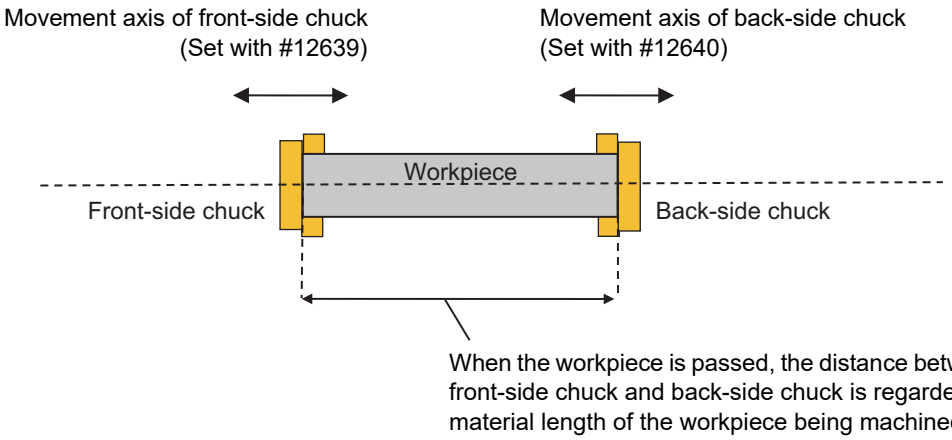
(9) If the value is rewritten by G10L70 (Parameter command by program), system variables or common variables in the program being checked, the value is rewritten in the same manner as when the program is executed during automatic operation.

(10) When M code specified by the parameters "#12631 main chuck close M" and "#12632 sub chuck close M" is executed, the workpiece is treated as handed to the front-side spindle or the back-side spindle.
When passing, the workpiece is treated as grabbed by the chuck at the position set in the parameters #12633 to #12637 (see the table below).

M code to be commanded	Delivery destination spindle of the workpiece	Destination of the work
M code specified to "#12631 main chuck close M"	Front-side spindle	Position specified by the following parameters #12633 main chuck pos X #12634 main chuck pos Y #12635 main chuck pos Z
M code specified to "#12632 sub chuck close M"	Back-side spindle	Position specified by the following parameters #12636 sub chuck pos X #12637 sub chuck pos Y #12638 sub chuck pos Z

Even when the workpiece is grabbed by both front- and back-side chuck in the actual machining, it is regarded to be grabbed by the chuck whose M code is commanded to close by simulation, shown in the table above.

(11) When the workpiece is passed, the chucks grab the workpiece regarding the distance between the front-side chuck and back-side chuck as the material length of the workpiece being machined. The distance between front-side chuck and back-side chuck is calculated as the axis position error set with the parameters "#12639 main chuck ax name" and "#12640 sub chuck ax name".



(12) When the sub part system control I/II commands are executed, the program error (P803) occurs and the check is stopped.

(13) During tap retract enable state, if the check operations ([Check continu] or [Check step]) is performed, "M01 Wait for tap retract 0057" is displayed and the check cannot performed in the part system in which the alarm occurred. After the state becomes tap retract enable and tap retract enable operation is completed, the check can be performed.

(14) When you use the arbitrary axis exchange control function, set all the part systems to be check targets for the graphic check (*1). When any axis in the part system which is not to be checked is designated, the operation error "M01 Arbitrary axis unexchangeable 1101" occurs, the graphic check cannot be performed in the part system where an alarm is displaying. Change the setting of the parameter or change the machining program.

(*1) The part system which is set to "0" by any of the parameters "#8982 CheckSmltns$1Invld" to "CheckSmltns$8Invld"

(15) It may take time to draw a helical interpolation path because it is drawn to the end point at a time.

4.4 3D Machining Simulation (M system) [M800VW Series]

Edit – 

3D machining simulation is a function to check the workpiece shape, tool movement, and interference of the cutting process without performing automatic operation. When any interference occurs, the contact area is displayed in the interference color set on the workpiece setting screen, and the program check is stopped.

This function is an optional specification.
Set the parameter "#19721 3Dmac.simu.inval" to "0" to use this function.
If the parameter is set to "1", the normal graphics check screen is displayed.

Press [Check] on the monitor or edit screen to switch to the 3D machining simulation check screen.
The main operations are the same as in the graphic check function.

[Check method]

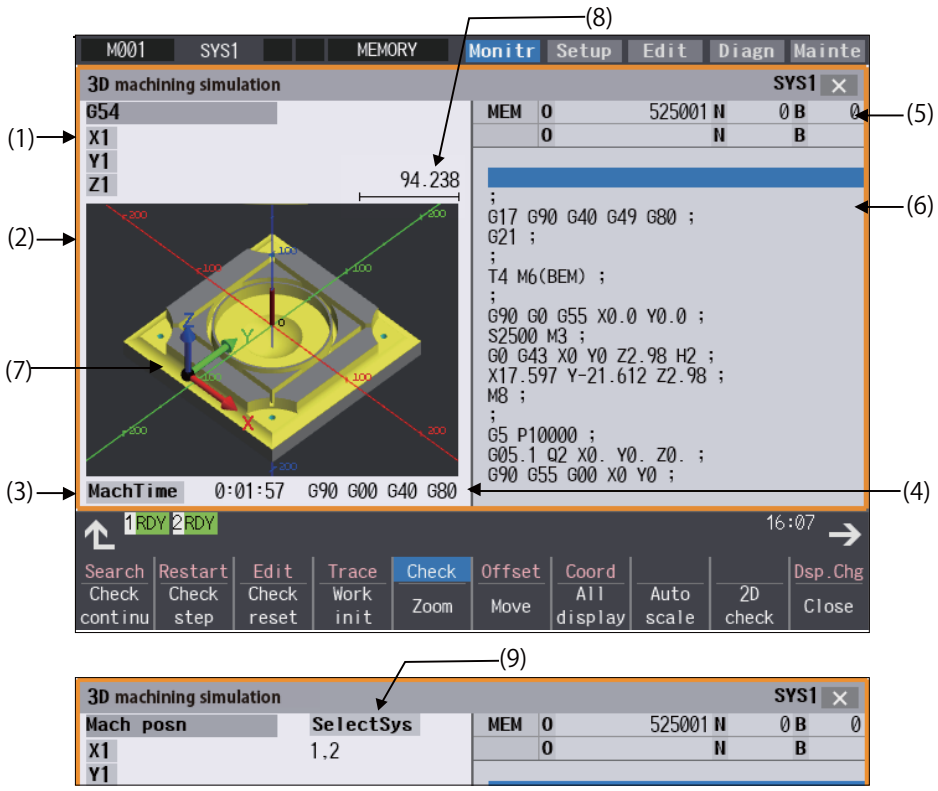
There are two types of 3D graphic checks as shown below, which can be switched between by pressing [Type change].

Note

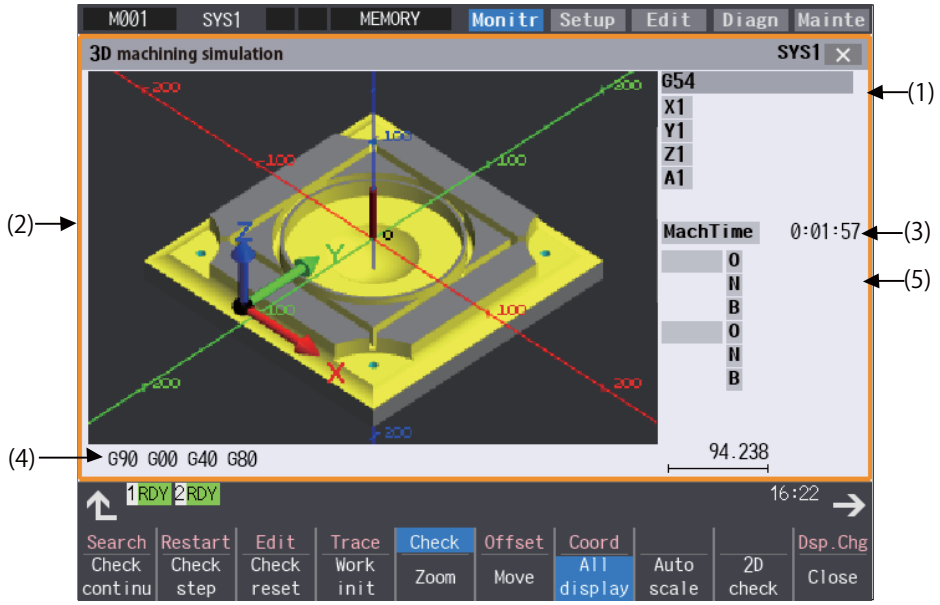
(1) Make sure to select the check method before the check is started. During the check, [Type change] will be grayed out, and the check method cannot be changed.

Check method	Outline
Check method I	This method simplifies the machining result of machining program and simulates at a high speed. Check method I will check machining programs in a single part system (displayed part system). In 3D drawing area, one workpiece and one tool are displayed.
Check method II	This method simulates the machining result of machining program at a higher accuracy than check method I. With check method II, machining programs in multiple part systems can be checked simultaneously. In 3D drawing area, one workpiece and tools for part systems are displayed. A simulation is run using the high-speed simple program check function. This method is available only when the operation mode in the part system to be simulated is in the memory mode. This method is operated in the machine lock status during the program check. Also, during the program check, the PLC signal (Graphic check (Check method II) in progress X759) is turned ON. The program check cannot be started during automatic operation in arbitrary part system. The program check is in operation from the time the program is executed by pressing either the [Check continu] or [Check step] key on the check screen until one of the following occurs: ♦Program ends. ♦The check reset key pressed ♦The reset signal or emergency stop signal is output ♦The screen switches from the check screen to another one

■Normal display mode ([All display] is OFF)



■Full-screen mode ([All display] is ON)



Display items

Display items			Details
(1)	Check counter		This displays the axis workpiece coordinate counter for the check. In normal display mode, the part system common axis name ("#1022 axname2") which conforms to the basic axis name set in the parameters "#1026 base_I", "#1027 base_J", and "#1028 base_K" will be displayed for the three axes to display. In the full-screen display mode, maximum 5 axes are displayed. Three axes are set by #1026, #1027 and #1028, and two axes are set by the rotary axis configuration parameters. When the rotary axis configuration parameters are not set, rotary axes set by the parameter "#1013 axname" are sequentially displayed. [Check method I] This displays the counter for the displayed part system (the part system being checked). The counter type will switch between the counter name and value for the machine position and work coordinate counter position, based on the drawing coordinate system in the workpiece shape setting screen. [Check method II] This displays the machine coordinate counter for the current part system.
(2)	3D drawing area		This displays the workpiece shape and tools for the cutting process. The drawing viewpoint can be moved, and the drawing size can be enlarged and reduced.
(3)	Machining time display		The time required for machining is calculated and displayed.
(4)	G modal display for check		The following items are displayed from the left. •Absolute/incremental (G90, G91: Group 3) •Operation mode (G00, G01, etc.: Group 1) •Tool compensation (G40, G41, G42: Group 7) •Fixed cycle (G70 etc.: Group 9) The items to be displayed vary depending on the status of group 24 command modal.
(5)	Currently executed machining program	Main O10...	This displays the device name, program No., sequence No., and block No. currently being executed. When they exceed the displayable number of characters (12 characters), the excess is abbreviated and displayed (*).
		Sub O1234…	This displays the sub-program device name, program number, sequence number, and block number for the sub-program currently being executed. When they exceed the displayable number of characters (12 characters), the excess is abbreviated and displayed (*).
(6)	Buffer display		This displays the contents of the machining program currently being executed. The block being executed is highlighted. When the parameter "#19431 Num display in prg" is set to "1", the variable, operation or function in the numerical part that follows each address is replaced with the calculated value and displayed for the current block and the next block.
(7)	Coordinate system display		This displays the coordinate system direction. This displays the axis direction of base axes I, J, K with arrows. The spacing of the graduations will vary depending on the workpiece size.
(8)	Scale display		This shows the display range scale. The unit is set to "mm/80pixel" in millimeters and "inch/80pixel" in inches.
(9)	Part system to be checked (Check method II only)		The part system to be checked is displayed.

Menus

Menus	Details
Check search	This performs a check search. Select a program to be checked from the program file list displayed in the pop-up window.
Check continu	This checks the program continuously.

Menus	Details
Check step	This checks the program per block.
Check reset	This resets the program check.
Work init	This initializes the workpiece shape. This displays the unmachined workpiece shape (memorized shape in NC).
Zoom	This enlarges/reduces the displayed workpiece shape.
Move	This moves the displayed workpiece shape.
All display	This switches the normal display mode and the full-screen display mode.
Auto scale	The scale is automatically set so that the unmachined workpiece shape set at the "Work set" window (workpiece shape setting) to be approximately 90% of the graphic area.
2D check	This changes the program check (2D) display. This menu cannot be selected while the check is performed or interrupted.
Colli check	This enables or disables the interference check. When interference check is enabled, if the tool contacts the machine or workpiece, the check stops. The contact portion is shown in a color highlighting the interference.
Opt Skip	Select whether to execute a block that has a slash "/" in the program to be checked. Selecting "ON" skips blocks with a slash "/" during the check. Selecting "OFF" also checks blocks with a slash "/". In check method II, [Opt Skip] is grayed out and non-selectable.
Rotate	This rotates the displayed workpiece shape.
Regist. Image	This associates a graphic check image with a machining program and save the program.
Scale display	This displays the scale of coordinate system direction and display range in the graphic drawing area.
M0/M1 neglect	[Check method I] If [M0/M1 neglect] is ON, program check does not stop even when M00/M01 is commanded. If [M0/M1 neglect] is OFF, program check stops when M00/M01 is commanded. [Check method II] The menu [M0/M1 neglect] is grayed out and non-selectable. The program check does not stops in the block where M00/M01 is commanded. (*1)
Simul. Mode	Switches between work simulation and machine simulation. If machine simulation is not available, this cannot be switched.
Type change	This switches between check method I and check method II. <Note> ♦When check method switch is invalid (parameter "#1231 set03/bit7" is set to "1"), this menu is not displayed.
Work setting	This sets the shape of the workpiece displayed during simulation.
Tool setting	This sets the tool shape used during simulation.

(*1) M code independent output (DM00/DM01) follows the parameter setting indicated below.

♦#1284 ext20/bit3 (M code output during multi-system simultaneous simulation (Check type II))
♦#1405 M_mode(SMLK) (M code output (during high-speed simple program check)
♦#1451 M[M031-000](SMLK) to #1466 M[M511-480](SMLK) (Special operation registration M code (High-speed simple program check))

Note

(1) The menus [Check search], [Check continu], [Check step], and [Check reset] are grayed out during automatic operation in trace mode, and the check is disabled.

(2) The menus [2D check], [Check search], [Work setting] and [Tool setting] cannot be selected while the check is performed or interrupted.

(3) [M0/M1 neglect] and [Regist. Image] are grayed out during continuous check and cannot be selected from the menu.

(4) The coordinate system direction and the scale display are held after the screen is switched, but they are not held after the power is turned OFF and ON again.

Sub-menu buttons for [Check search]

Menus	Details
Memory	This selects the device for searching for the program. When a device with directory is selected, the root is selected first.
Memory2	
HD	
Memory card	
DS	
USB Memory	
Top jump	Display starts from the first page of the directory/list of files. The cursor moves to the top line.
Bottom jump	The last page of the directory/list of files is displayed. The cursor moves to the bottom line.
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.
Preview	This switches to show or hide the preview (detailed information).
Change Disp	This switches between list display mode and thumbnail display mode.
List update	This updates the list contents. (The latest contents of the currently selected device and directory are listed.)
Sort change	This changes the method of sorting the list.
Close	This closes the pop-up window and quits this function.

4.4.1 Switching Check Method

Refer to "4.2.1 Switching Check Method".

4.4.2 Checking Continuously

Refer to "4.2.2 Checking Continuously".

4.4.3 Checking Block by Block

Refer to "4.2.3 Checking One Block at a Time".

4.4.4 Checking Multiple Part Systems Simultaneously

Refer to "4.2.4 Checking Multiple Part Systems Simultaneously".

4.4.5 Canceling the Program Check

Refer to "4.2.5 Canceling the Program Check".

4.4.6 Enlarging/Reducing the Workpiece Shape

Refer to "4.2.6 Enlarging and Reducing the Workpiece Shape".

4.4.7 Moving the Workpiece Shape

Refer to "4.2.7 Moving the Workpiece Shape".

4.4.8 Rotating the Workpiece Shape

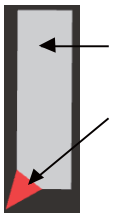
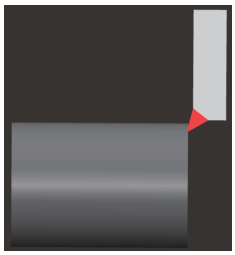
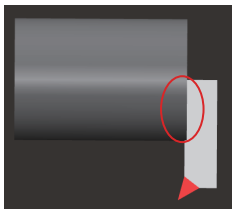
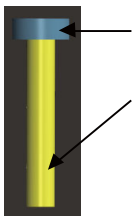
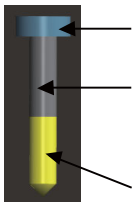
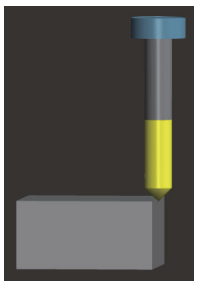
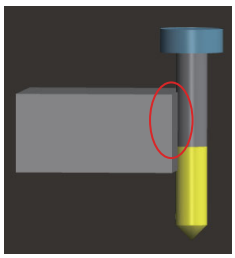
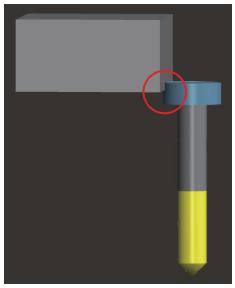
Operation method

(1) Press the menu [Rotate].

(2) Use the menus [Up], [Down], [Left], [Right] to adjust the display angle.
The cursor movement keys ([↑], [↓], [←], [→]) can also be used for moving.

4.4.9 Performing an Interference Check

Simulations allow you to check for tool, workpiece, or mechanical part interference before actual machining.
When there is contact between the tool, workpiece, and/or a mechanical part, the contact area is displayed in color set for interference on the workpiece setting screen, and the program check is stopped.
If machine simulation is not available, this checks for interference between the non-cutting parts of the tool and the workpiece.

Tool	Example of non-interference	Example of interference
Turning tools  	 The tool nose is in contact with the workpiece	 The shank is in contact with the workpiece
Tapping tool  Other mill tools  	 The tool nose is in contact with the workpiece	 The shank is in contact with the workpiece  Holder is in contact with the workpiece

Note

(1) If the tool contacts something other than the workpiece, the check stops due to interference, regardless of where the tool is in contact.

Operation method

(1)	Press the menu [Colli check]. Press [Colli check] again to unhighlight it and deactivate interference check.	→	The menu button is highlighted.
(2)	Press the menu [Check step] or [Check continu].	→	The program check is executed, and workpiece shape in the cutting process is drawn. A "Checking" message is displayed and [Check step] or [Check continu] is highlighted. If the parts subject to the interference check come in contact, the parts are shown in a color highlighting the interference.

4.4.10 Setting the Workpiece Shape

Refer to "4.2.10 Setting the Workpiece Shape".

4.4.11 Setting the Tool Shape

Press the menu [Tool setting] to display the tool management screen. Set the tool shape used in the solid display for graphic check on the tool management screen.

Even though the menu [T-Mng.] is pressed on the [Setup] screen, the same screen can be displayed. For details, refer to "6.5 Setting the Tool Management Data (Tool Management Screen)".

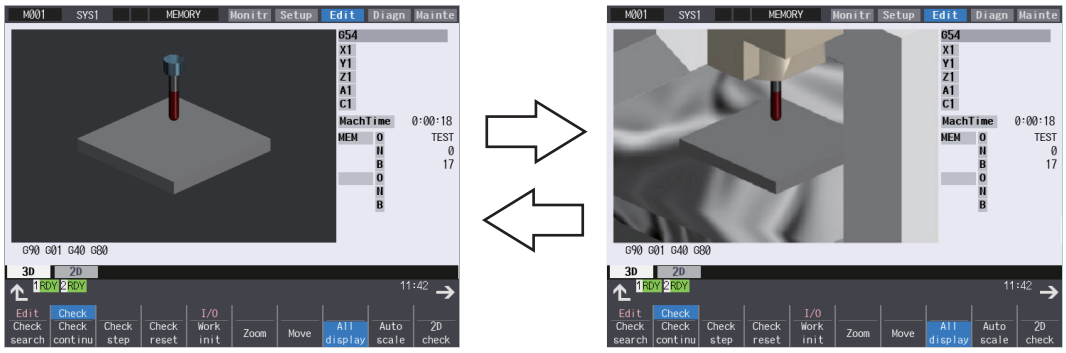
4.4.12 Associating Graphic Check Images with a Machining Program

Refer to "4.1.14 Associating Graphic Check Images with a Machining Program".

4.4.13 Switching Simulation Modes

Press [Simul. Mode] to switch between machine simulation and workpiece simulation.

Machine simulation	The workpiece and tool are shown along with the machine and jig, allowing the actual relation to the machine position and the interference position to be determined.
Work simulation	Shows the tools and workpiece in isolation, making it easier to see the machining workpiece shape and check on the machining process.



Work simulation

Machine simulation

Note

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

(2) The simulation mode is applied commonly to all part systems.

(3) If machine simulation is not available and [Simul. Mode] is pressed, a message will display and the system will not switch to machine simulation.

(4) If the parameter "#11805 Machine Simu Model" is set to "0", [Simul. Mode] is grayed out and cannot be selected.

(5) Even in work simulation mode, interference will result and the check will stop if the machine or jig come in contact.

4.4.14 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	A mirror image is applied and a check drawing is made at the setting status prior to starting the check.
Mirror image by external input	[Check method I] Signals are ignored and a check drawing is made. [Check method II] Signals are reflected and a check is performed.
Z axis command cancel	Signals are ignored and a check is performed.
Interlock	Signals are ignored and a check is performed.
External deceleration	Signals are ignored and a check is performed.
Override	Signals are ignored and a check is performed. A check drawing is performed with override fixed at 100% ignoring the signals (check method II).
Auto operation pause (feed hold)	Signals are ignored and a check drawing is made.
Auto operation start-up (cycle start)	[Check method I] Operation can be started automatically even during a check (background). [Check method II] If the cycle start button is pressed during checking, the operation alarm "M01 program check mode" is displayed while the button is held down.
Auto-restart	The check is completed at M02/M30 and is not restarted.
Single block	Signals are ignored and a check is performed.
Manual arbitrary reverse run	Signals are ignored and a check is performed.
High-speed simple program check	Signals are ignored and a check is performed.
Tool life management	The tool life data count is not performed.
Skip (G31)	The skip signal is disabled. A drawing moves to the end point and is completed.
Workpiece coordinate offset	The offset is enabled.
Manual mode/handle	[Check method I] During checking, you can switch to the manual mode, HANDLE, to move the machine. However, the drawing position of the check will not be affected by the mode change (background). [Check method II] If the mode is switched to the manual mode during checking, the operation alarm "M01 program check mode" is displayed (foreground).
Tape operation and MDI operation	If the mode is switched to the manual mode during checking, the operation alarm "M01 program check mode" is displayed (check method II).
F 1-digit feed	The speed does not affect the check.
User macro I/II	Basic variable calculations, decisions and branch functions are all enabled. However, the following functions are ignored. •Macro interface input/output •Single block stop •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

Function name	Remarks
M, S, T and 2nd miscellaneous function commands	M, S, T and 2nd miscellaneous function command signals commanded by the program during the check are not output. Neither do they display on the operation screen etc. The auxiliary function command by manual operation on the position display screen is output even during checking. With the setting of the parameter "#1284 ext20/bit3", not outputting/outputting the miscellaneous function can be switched (check method II).
G command modal	G modal displays on the modal display screen while the check drawing is being made. The modal information after the check cannot be returned to the state before the check (check method II).
Optional block skip	The setting status used when starting the drawing check (Check method II). Use the menu to switch ON and OFF. The optional block skip state is held until the power is turned ON again, which is common to part systems. However, when the power is turned ON again, the mode is always turned OFF (Check method I).
Spindle position control (spindle/C axis control)	Set the spindle position control (spindle/C axis control) for the part system to be checked to spindle mode, then execute a program check. Executing the check in C axis mode causes the operation error "M01 SP-C ax ctrl runs indepndntly 1026", and the check cannot be started (Check method II). When the program command method is selected, the spindle/C axis moves in C axis mode (Check method I).

Note

(1) Reference position return and start position return
The reference position return command and start position return command are enabled, but actual machine motion is partially different from the simulated tool movement.
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.
However, when the tool is moved by simulation, it moves from the intermediate point to the origin at the same time on all axes. Also, in check method I, reference position return moves to the position set in parameter "#8217 Check start point". When this happens, set the reference position to parameter #8217.

4.4.15 Handling of Variable Commands, Programmable Parameter Input, and Programmable Compensation Data Input

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.

Meaning of symbols in the table
○: The value returns to the status prior to starting the program check (Actual machining data is not rewritten).
×: The value does not return to the status prior to starting the program check (Actual machining data is rewritten).

■Check method I

Data type	Data status	Explanation
Parameters	○	The programmable parameter input commands which have been executed at the program check are not set as actual data.
Local variables	○	"#1231 set03/bit0" (Graphic check compatibility parameter) is disabled.
Common variables	○	
Workpiece coordinate system offset	○	
Tool compensation amount	○	

■Check method II

Data type	Data status	Explanation
Parameters	×	The programmable parameter input commands which have been executed at the program check are set as actual data.
Local variables	×	"#1231 set03/bit0" (Graphic check compatibility parameter) is disabled.
Common variables	×	
Workpiece coordinate system offset	×	
Tool compensation amount	×	

4.4.16 Precautions for 3D Machining Simulation

(1) The check function is valid for only NC internal operation results. Thus, the path is not drawn for the command which requires machine movement.
(2) For the handling of various types of data being set in the program, refer to "4.4.15 Handling of Variable Commands, Programmable Parameter Input, and Programmable Compensation Data Input".
(3) "Program check completed" appears with M02/M30 command.
(4) Graphic check is not available for serial and MDI program.
(5) Check search is not available for the program whose file name is consist of 33 characters or more.
(6) The drawing coordinate system selections for 3D machining simulation and 3D solid program check are linked, but settings for the drawing coordinate system for 3D machining simulation and 2D graphic check are not linked.
(7) If the screen is switched during checking, the check reset occurs and the check is interrupted.
(8) If tracing is turned ON during automatic operation, [Check search], [Check continu], [Check step], [Check reset] and other options are grayed out. Check cannot be performed by 3D machining simulation.
(9) The workpiece shape is not initialized even if an operation such as enlarge, reduce, move, change display mode, or rotate is set during simulation. Also, the program does not stop.
(10) With check method I, if the part system is switched during checking, the check is reset.

(11) If the following data of the tool corresponding to the T number is out of range when a tool change command is issued, the default value is displayed.
If the shape data of the tool corresponding to the T number is invalid when the tool is changed, it is displayed as a default tool.
Refer to the table below for detailed conditions.

Type	Shape data	Default value	Out-of-range conditions	Invalid conditions
Turning	Tool length A	150 mm	Less than 2.000, more than 9999.999	-
	Tool length B	25 mm	Less than 2.000, more than 9999.999	Less than "Nose R"× (1.0 + sin ("Cutting edge angle" - 90.0°))
	Nose R	1 mm	Less than 0.000, more than 9999.999	-
	Nose angle	60°	Less than 1.000, more than 179.999	More than 89.999
	Cutting edge angle	95°	Less than 0.001, more than 179.999	-
	Tool width	27.5 mm	Less than 2.000, more than 9999.999	"Tool length B" or less
	Installation angle	0°	Less than -359.000, more than 360.000	-
	GangTool ofs Y	0 mm	-	-
Groove	Tool length A	150 mm	Less than 2.000, more than 9999.999	"Maximum groove depth"× 1.01 or less
	Tool length B	25 mm	Less than 2.000, more than 9999.999	"Nose width" or less
	Nose R	1 mm	Less than 0.000, more than 9999.999	-
	Nose width	3 mm	Less than 2.000, more than 9999.999	"Nose R"× 2 or less
	Maximum groove depth	12 mm	Less than 2.000, more than 9999.999	"Nose R"× 2 or less
	Tool width	27.5 mm	Less than 2.000, more than 9999.999	"Tool length B" or less
	Installation angle	0°	Less than -359.000, more than 360.000	-
	GangTool ofs Y	0 mm	-	-
Thread cutting	Tool length A	150 mm	Less than 2.000, more than 9999.999	(("Tool length B") × tan (90.0° - (45.0° / 2.0))) or less
	Tool length B	25 mm	Less than 2.000, more than 9999.999	-
	Tool width	27.5 mm	Less than 2.000, more than 9999.999	Less than "Tool length B"
	Installation angle	0°	Less than -359.000, more than 360.000	-
	GangTool ofs Y	0 mm	-	-
Drill/Lathing drill	Length	150 mm	Less than 2.000, more than 9999.999	Less than ("Tool radius × tan (90.0° - ("Nose angle" / 2.0))) + "Holder H"
	Diameter	20 mm	Less than 2.000, more than 9999.999	-
	Nose angle	118°	Less than 1.000, more than 179.999	-
	Blade length	50 mm	Less than 2.000, more than 9999.999	-
	Holder H	50 mm	Less than 0.000, more than 9999.999	-
	Holder D	30 mm	Less than 0.000, more than 9999.999	-
	Installation angle	0°	Less than -359.000, more than 360.000	-
	GangTool ofs Y	0 mm	-	-

Type	Shape data	Default value	Out-of-range conditions	Invalid conditions
Tap/Lathing tap	Length	150 mm	Less than 2.000, more than 9999.999	-
	Diameter	20 mm	Less than 2.000, more than 9999.999	-
	Pitch	5 mm	Less than 2.000, more than 9999.999	More than "Length"
	Root diameter	0.05 mm	Less than 2.000, more than 9999.999	-
	Holder H	50 mm	Less than 0.000, more than 9999.999	-
	Holder D	30 mm	Less than 0.000, more than 9999.999	-
	Installation angle	0°	Less than -359.000, more than 360.000	-
	GangTool ofs Y	0 mm	-	-
Tap unified thread/tap pipe thread/ turning unified thread/ turning pipe thread	Length	150 mm	Less than 2.000, more than 9999.999	-
	Diameter	20 mm	Less than 2.000, more than 9999.999	-
	Holder H	50 mm	Less than 0.000, more than 9999.999	-
	Holder D	30 mm	Less than 0.000, more than 9999.999	-
	Installation angle	0°	Less than -359.000, more than 360.000	-
	GangTool ofs Y	0 mm	-	-
Ball end mill/ flat end mill	Length	150 mm	Less than 2.000, more than 9999.999	-
	Diameter	20 mm	Less than 2.000, more than 9999.999	-
	Blade length	50 mm	Less than 2.000, more than 9999.999	-
	Holder H	50 mm	Less than 0.000, more than 9999.999	-
	Holder D	30 mm	Less than 0.000, more than 9999.999	-
	Installation angle	0°	Less than -359.000, more than 360.000	-
	GangTool ofs Y	0 mm	-	-
Radius end mill	Length	150 mm	Less than 2.000, more than 9999.999	Less than "Corner R" + "Holder H"
	Diameter	20 mm	Less than 2.000, more than 9999.999	-
	Corner R	1 mm	Less than 2.000, more than 9999.999	More than tool radius
	Blade length	50 mm	Less than 2.000, more than 9999.999	-
	Holder H	50 mm	Less than 0.000, more than 9999.999	-
	Holder D	30 mm	Less than 0.000, more than 9999.999	-
	Installation angle	0°	Less than -359.000, more than 360.000	-
	GangTool ofs Y	0 mm	-	-
Chamfer	Length	150 mm	Less than 2.000, more than 9999.999	Less than (("Tool radius" - "Tip radius")× tan ("Tip angle")) + "Holder H"
	Diameter	20 mm	Less than 2.000, more than 9999.999	-
	Tip angle (*1)	atan (2.0 / ("Diameter" - "Tip dia."))	Less than 1.000, more than 179.999	89.9 or more
	Tip dia. (*2)	5 mm	Less than 2.000, more than 9999.999	-
	Blade length	50 mm	Less than 2.000, more than 9999.999	-
	Holder H	50 mm	Less than 0.000, more than 9999.999	-
	Holder D	30 mm	Less than 0.000, more than 9999.999	-
	Installation angle	0°	Less than -359.000, more than 360.000	-
	GangTool ofs Y	0 mm	-	-

Type	Shape data	Default value	Out-of-range conditions	Invalid conditions
Face mill	Length	150 mm	Less than 2.000, more than 9999.999	-
	Diameter	20 mm	Less than 2.000, more than 9999.999	-
	Cutter L	50 mm	Less than 2.000, more than 9999.999	More than "Length", "Tool diameter" / tan (0.1°) or more, "Tool diameter" / tan (89°) or less
	Shank dia.	10 mm	Less than 2.000, more than 9999.999	-
	Holder H	50 mm	Less than 0.000, more than 9999.999	-
	Holder D	15 mm	Less than 0.000, more than 9999.999	-
	Installation angle	0°	Less than -359.000, more than 360.000	-
	GangTool ofs Y	0 mm	-	-
Boring/back boring	Length	150 mm	Less than 2.000, more than 9999.999	Less than ("Tool radius" / 2.0) + "Holder H"
	Diameter	20 mm	Less than 2.000, more than 9999.999	-
	Holder H	50 mm	Less than 0.000, more than 9999.999	-
	Holder D	30 mm	Less than 0.000, more than 9999.999	-
	Installation angle	0°	Less than -359.000, more than 360.000	-
	GangTool ofs Y	0 mm	-	-

(*1) When the "Tip angle" value of chamfering tool is out of the range, atan (2.0 / ("Diameter" - "Tip dia.")) is set instead of the default value.

(*2) When the "Tip dia." value of chamfering tool is equal to or larger than "Diameter", the "Tip dia." value is set to a half of "Diameter".

(12) If the tool data is inconsistent when a tool change command is issued and the tool cannot be displayed, the drill tool is shown.

(13) If the blade length is longer than the tool length, or if the blade length is 0, the tool will be displayed assuming that blade length and tool length are the same.

(14) If the sum of the blade length and holder height is greater than the tool length, the holder height will be treated as 0 mm.

(15) If the parameter "#8920 3D tool ofs select" is set to "0" or "1" and the displayed tool radius is 0 or less, the 20-mm default tool radius is assumed and drawn.

(16) With check method I, the workpiece coordinate position will not be changed even with a coordinate system switching command included in the program executing the program check. Therefore, the drawings may display as overlapped.

(17) The check search cannot be executed when the sub part system that cannot be operated is displayed. If the search is carried out, the message "Search error" appears.

(18) If a travel command is executed during polar coordinate interpolation or cylindrical interpolation on a machine which cannot perform C axis control on its workpiece spindle, the tool will not move.

(19) On machines with a Y-axis, if polar coordinate interpolation or circular interpolation is executed while the Y-axis is not at 0 in the programming coordinate system, the tool will move differently from the actual machining.

(20) When the workpiece shape is changed from the external source such as another application, the drawing reflect to the workpiece shape on the 3D check screen by pressing the [Work setting] menu to display workpiece shape setting screen.

Precautions for check method I

(1) The program that you have selected during Check Search is not the program that is executed during automatic operation. Search the program for automatic operation by using the operation search function of the monitor screen.

(2) The parameters and compensation amount do not reflect on check drawing even if you set them during checking. To reflect them on the drawings, reset the check once. Then, press the menu [Check continu] or [Check step] and execute the check again.

(3) "Program check completed" appears with M02/M30 command.

(4) The check is not terminated via execution of a reset.
If a program error occurs, release the error by selecting the menu [Check reset].

(5) During check search, if the length of the full path name exceeds 48 characters, you cannot move to another directory.

(6) If the menu [Opt skip] is ON (highlighted), the blocks that have the "/" character are skipped and the T-code list is obtained.

(7) The file name continues to be displayed as an O-number display even if you erase or rename the file after it is checked and searched. If you check a file in this status, the "Push [Check reset] menu" message is displayed.

(8) During checking, if you read the integrated time by using variable number #3001 or #3002, the checking start time is obtained. You cannot write the integrated time.

(9) When [Check continu] is executed, even if variable number #3006 is used for checking, you cannot stop the program and you cannot display the message.

(10) When the M01 is commanded with the menu [M0/M1 neglect] disabled, the drawing stops regardless of the state (ON or OFF) of the optional stop switch.

Notes for check method II

(1) If you press the cycle start button when an arbitrary part system is being checked, the operation alarm "M01 program check mode" is generated and the cycle start is disabled while the button is held down.
Press the auto start button after the check is completed or after the check is stopped with the menu [Check reset].

(2) If you carry out the operation search or check operation (continuous or step check) during cycle start or automatic stop of an arbitrary part system, an operation message "Executing automatic operation" is displayed, and no search or check drawing is carried out.

(3) "Program check completed" appears with M02/M30 command. For the tape mode, the rewind process is not carried out.

(4) During automatic operation, the contents of the program under automatic operation are displayed by the "Currently executed machining program" (ONB display) and "Buffer display" on the 3D machining simulation screen. During checking, the contents of the checking program are displayed by the "Currently executed machining program" (ONB display) and "Buffer display" on the monitor screen.

(5) If you carry out the operation search or check operation (continuous or step check) when the program is restarted, the "Program restarting" operation message is displayed, and no search or check drawing is carried out.

(6) If the manual operation mode is selected during checking, the "M01 program check mode" is displayed and the check is stopped. When the mode returns to the memory mode again, the alarm is canceled and the check can be carried on.

(7) When the PLC Interrupt signal is turned ON during checking, the "M01 Program check mode" is selected. To start the PLC interrupt, terminate the current check by executing a reset. When you turn the PLC interrupt signal OFF, the alarm is released. You can continue the current checking even if the PLC Interrupt signal is ON.

(8) If the value is rewritten by G10L70 (parameter command by program), system variables or common variables in the program being checked, the value is rewritten in the same manner as when the program is executed during automatic operation.

(9) When the sub part system control I/II commands are executed, the program error (P803) occurs and the check is stopped.

(10) During tap retract enable state, if the check operations ([Check continu] or [Check step]) is performed, "M01 Wait for tap retract 0057" is displayed and the check cannot performed in the part system in which the alarm occurred. After the state becomes tap retract enable and tap retract enable operation is completed, the check can be performed.

(11) When using the arbitrary axis exchange control function, set all the part systems to be checked for the graphic check (*1). When any axis in the part system which is not to be checked is designated, the operation error "M01 Arbitrary axis unexchangeable 1101" occurs, the graphic check cannot be performed in the part system where an alarm is displaying. Change the setting of the parameter or change the machining program.

(*1) The part system which is set to "0" by any of the parameters "#8982 CheckSmltns$1Invld" to "#8989 CheckSmltns$8Invld".

Setting for base axes

When a rotary axis is included among the base axis parameters I, J, and K (#1026 to #1028), set the base axis parameters for graphics I, J, and K (#8631 to #8633) to exclude the rotary axis as the following example.

If all the base axis parameters for graphics I, J, and K are not set, the graphic check operates in accordance with the setting of the base axis parameters #1026 to #1028.

[Setting example of parameters]

Base axis parameter

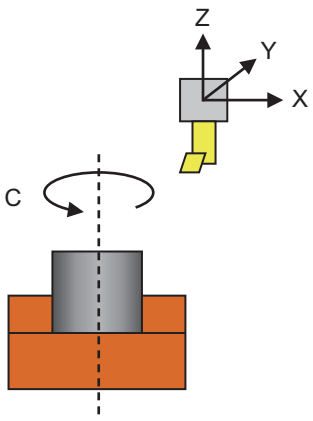
No.	Parameter name	Setting value	Explanation
#1026	base_I	X	Linear axis
#1027	base_J	C	Rotary axis
#1028	base_K	Z	Linear axis

Base axis parameter for graphics

No.	Parameter name	Setting value	Explanation
#8631	GraphicsBaseAxis_I	X	Set the same axis name as #1026.
#8632	GraphicsBaseAxis_J		Enter "0" to the rotary axis to be blank.
#8633	GraphicsBaseAxis_K	Z	Set the same axis name as #1028.

Using the lathe turning tool

The graphic check using a lathe turning tool is applicable only to the machine configuration with the C axis as the spindle.



Tool position and shape

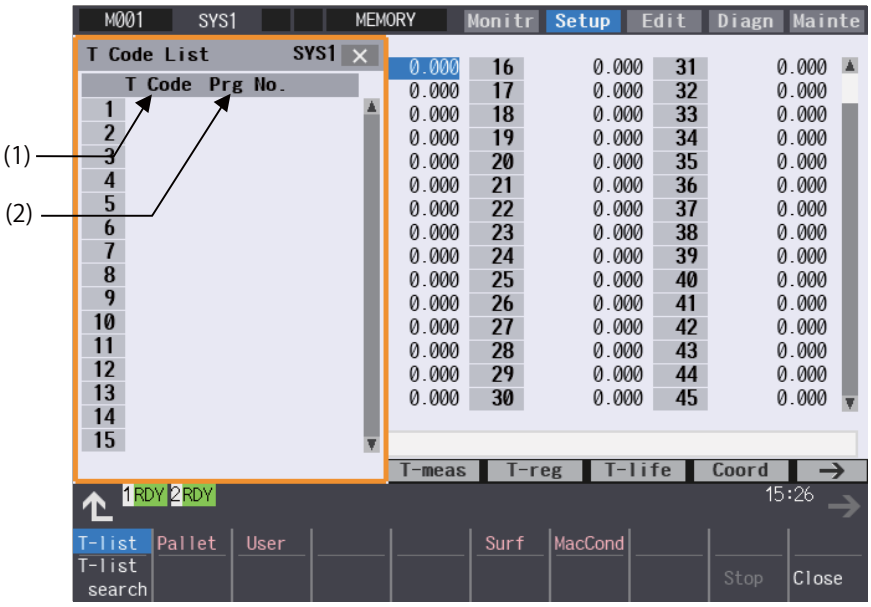
Set the tool position with the parameter "#8920 3D tool ofs select" according to the tool length measurement type (I/II). For details on the tool position and the diameter and length of the tool to be displayed based on the combination with the tool length measurement type (I/II), refer to "drawing position" in "4.2.17 Notes for Graphic Check (3D)".

4.5 Checking the Tool Command (T Code List)

Setup – T-list (M system)

Setup – T-list (L system)

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.
Refer to "6.2 Registering a Tool (Tool Registration Screen)" for details on registering a tool.



Display items

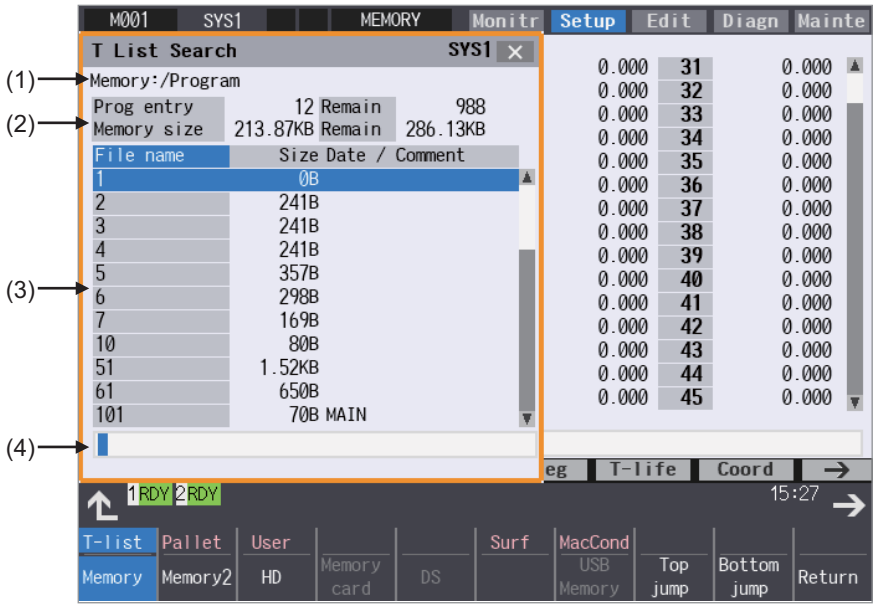
Display items	Details
(1) T code	T codes are listed in the order that the T code is commanded in the designated machining program (including subprogram). (As for the T code of the tool that has not been registered, it is indicated with text color and background color changed.)
(2) Prg No.	This shows designated program No. When the number of characters exceeds 15, the excess is indicated as "*".

Menus

Menus	Details
T-list search	This designates the program. This menu is invalid while getting the T codes.
Stop	This interrupts getting the T codes. This menu is valid only while getting the T codes.
Close	This closes the pop-up window and quits this function.

4.5.1 Searching T List

Press the menu [T-list search] to display the following screen.



Display items

	Display items	Details
(1)	Device name, directory display	This displays the device and directory where the searched machining program is located.
(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 PAGE and PAGE to scroll the displayed list. <Displaying "Date/comment" field> When the device is HD, memory card, DS or USB memory, the file update date/time is displayed. When NC memory or NC memory 2 is selected as a device, the display can be switched as follows depending on the parameter "#8981 NCmemory date disp" setting: (*1) When #8981 is set to "0", the machining program comment is displayed. When #8981 is set to "1", the file update date/time is displayed. Whether to show or hide the comment field can be selected by pressing the menu [Comment nondisp]. 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 section	This displays details of the key input.

(*1) For the program list view of the NC memory or NC memory 2, the program comment is displayed when the multi-part system program management is valid (#1285 ext21/bit0 = 1), regardless of the parameter setting of "8981 NCmemory date disp".

Menus

Menus	Details
Memory	This selects the device for searching for the program. When a device with directory is selected, the root directory is selected first. This menu [HD] is not displayed on M800VS/M80V Series. (*1)
Memory2	
HD	
Memory card	
DS	
USB Memory	
Top jump	Display starts from the first page of the directory/list of files. The cursor moves to the top line.
Bottom jump	The last page of the directory/list of files is displayed. The cursor moves to the bottom line.
Comment nondisp	This hides the comment field in the list. The file name field can be enlarged.
List update	This updates the list contents. The latest contents of the currently selected device and directory are listed.
Sort change	This changes the method that the list is sorted.
Return	This closes the pop-up window and starts getting T codes.

(*1) When the HD mode operation on an industrial computer (IPC) is enabled with M80V Series, the menu [HD] becomes selectable and you can select files in the hard disk on the industrial computer. The availability of this function depends on the MTB specifications (parameter "#1760 cfgPR10/bit2").
Set the IP address of the industrial computer to the parameter #11005 to use this function.

Displaying the T code list

(1) Press the menu [T-list] on Setup screen.

(2) Press the menu [T-list search].

(3) Select the device.
(Example) Memory

(4) For devices other than NC memory and NC memory 2, the root directory is selected.
To move to another directory, use the [↑], [↓], PAGE, or PAGE key to align the cursor with the directory of target file, and press the [INPUT] key.
<Note>
- If the list contents differ from the actual device or directory, press the menu [List update].
- Press the menu [Sort change] to change the sort. Sorting method changes to File name ascending order -> File name descending order -> Date/Comment ascending order -> Date/Comment descending order -> File name ascending order -> ..... every time the menu key is pressed.

(5) Use the [↑], [↓], PAGE, or PAGE key to align the cursor with the target machining program, and press the [INPUT] key. ➡ T code list appears as a pop-up window.
Getting T code and listing starts.
The menu [Stop] becomes valid.
The [T-list search] is invalid.

T code list cannot be displayed when [INPUT] key is pressed with the following programs selected:
- The program that does not exist.
- The program being checked.

For M80V series, T code list cannot be displayed when [INPUT] key is pressed with the following programs selected:
- The program during automatic operation.
- The program that is waiting to be restarted. The list window will not change.

Note

(1) When switching to the other screen (monitor screen or edit screen), getting T code is interrupted automatically.
(2) When the menu [Close] is pressed, getting T code is interrupted automatically.
(3) For some models, T list search cannot be carried out with manual operation mode (except memory, tape, online, MDI, PC or direct mode). Switch it to automatic operation mode.
(4) T list search is not available for sub part system I/II for M80V Series. If the search is carried out, the message "Search error" appears.
(5) T list search is not available for sub part system II for M800V Series. If the search is carried out, the message "Search error" appears.
(6) T code list for MDI program cannot be displayed.
(7) T code list of a program with 33 or more file name characters cannot be gotten.
(8) When you designate the program which occurs a program error, T code list can be gotten up to the previous block of the error. Error message is not displayed.

Stopping the update of T code list

It may take time to update the T code list. When the menu [Stop] is pressed while the T code list display is updated, the acquisition of the T code list is stopped.
In this case, the T code list up to the position where it was stopped can be displayed.

5

Inputting and Outputting a Machining Program

This chapter explains the details of Inputting and outputting the machining program mainly based on the menu [I/O] on the Edit screen.
This operation can be performed with the menu [I/O] on the Maintenance (Mainte) screen as well.

5.1 Inputting and Outputting a Program

Edit – 

The machining program can input/output between the NC internal memory and the external input/output device on the Edit screen. The built-in disk of the display unit inside the NC unit is also treated as an external device.

Note

(1) When the program input/output function is used, set the parameter "#8923 Hide Edit-IO menu" to "0".

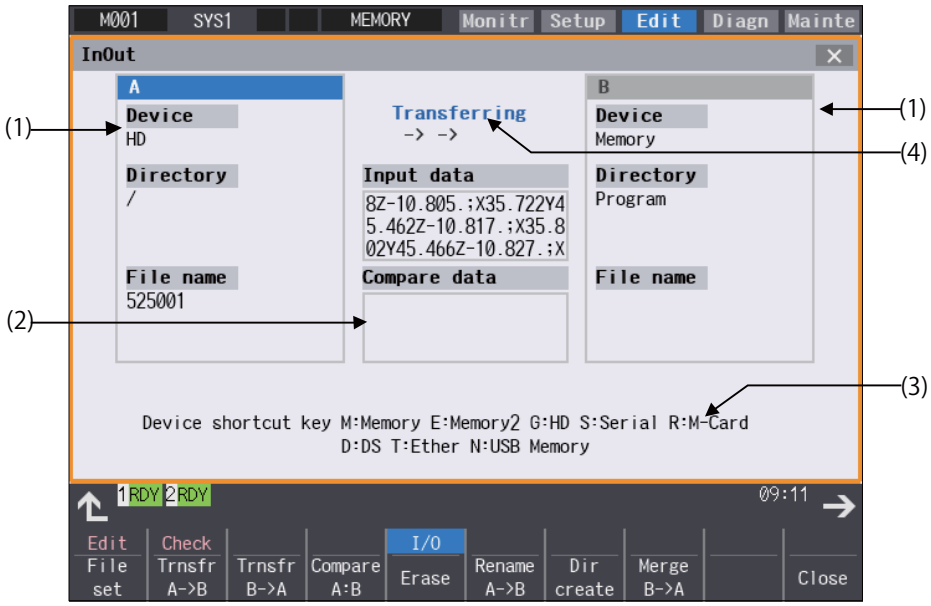
This function applies to the following devices.

- NC memory (SRAM memory at NC)
- NC memory 2 (built-in SD card)
- HD
- Serial
- Memory card (front-side SD card)
- DS (control unit-side SD card)
- Ethernet
- USB memory

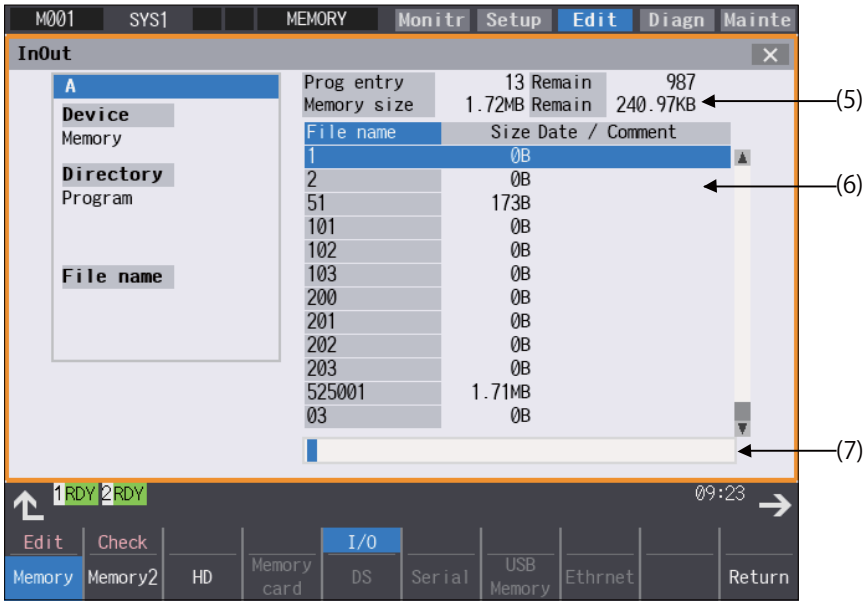
[Inputting and outputting of multi-part system program]

When the parameter "#1285 ext21/bit0" (Multi-part system program management) is set to "1" and NC Memory or NC memory 2 is selected as a device, a program input and output will be executed across all part systems in batch if a parameter "#1285 ext21/bit2" (Multi-part system program generation and operation) is set to "0". When it is set to "1", the program input and output will be executed for the program of displayed part system.
However, when the machining program is transferred from the NC memory to the other device, the transfer will be executed across all part systems in batch if "#1286 ext22/bit5" (Selection of multi-part system program input/output method) is set to "0". If it is set to "1", only displayed part system's program will be transferred.

<During file transmission>



<During file setting>



Display items

Display items		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 in three lines, file name: 63 in three lines), the excess is not displayed. As for the device and directory, the setting values are retained even when the power is turned OFF. When a medium is changed on a device where a USB memory device, etc. can be removed or inserted, the directory before the change is displayed. In this case, press the menu [File set], input "/", then move to the target directory. When multiple files are selected, the selected file name and "...etc" is displayed in the file name setting area.

Display items		Details
(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 is displayed.
(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 (*1)(*2)	This displays the file (machining program) registration count information and memory capacity information for the selected device. Prog entry: Displays the number of programs that are already registered as the user's machining programs. Remain: This displays the remaining number of programs that can be registered when NC memory or NC memory 2 is selected as a device. The total of the number of programs registered and the remainder is the maximum number of registrations set in the specifications. Memory size: Displays the capacity that is already registered as the user's machining programs. Remain: Displays the remaining capacity that can be registered. When NC memory or NC memory 2 is selected as a device, the total of the memory capacity and the remaining capacity is used as the maximum memory capacity defined in the specifications.
(6)	List display section (*3)	This displays the directory contents list (directory and file name) of the device A or B where the cursor is currently located. When the menu [Comment nondisp] is highlighted, the "Date/Comment" column is not displayed. Up to 32 characters are displayed in the file name column. File name: ♦When NC memory or NC memory 2 is selected as a 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 NC memory or NC memory 2 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 the maximum (13 characters with the comment column, 32 characters without the comment column), the excess is indicated as "*". Size: ♦This displays the size of each file. (The number of characters in the machining program when NC memory or NC memory 2 is selected for the device.) ♦When directory is selected, this displays "DIR". Date/Comment: ♦This displays the comment (up to 19 alphanumeric characters and symbols) of each file. ♦The date which the file is updated is displayed for the HD, memory card, DS, USB memory or Ethernet. ♦For NC memory or NC memory 2, the display can be switched as follows depending on the parameter "#8981 NCmemory date disp" setting: (*4) When #8981 is set to "0", the machining program comment is displayed. When #8981 is set to "1", the file update date/time is displayed. ♦When the comment of the NC memory or NC memory 2 exceeds the number of display characters (19 characters), the exceeded part is not displayed. The excess is not abbreviated to "*".
(7)	Input section	This displays the input key.

(*1) Some items may not be displayed depending on the device.
○: Displayed ×: Not displayed

Display item/Device	Memory/ Memory2	HD	Serial	Memory card	DS/USB Memory	Ethernet
Number of programs registered	○	○	×	○	○	○
Remain	○	×	×	×	×	×
Number of memory characters	○	○	×	○	○	○ *
Remain	○	○	×	○	○	×
List	○	○	×	○	○	○

* When the parameter "#97*1 Host n no total siz" is set to "1", the number of host memory characters will not appear.

(*2) When the device is not mounted, a registration count and a memory capacity are not displayed and an empty list is displayed.

(*3) When the device is serial, an empty list is displayed.

(*4) For the program list view of the NC memory, the program comment is displayed when the multi-part system program management is valid (#1285 ext21/bit0 = 1), regardless of the parameter setting of "8981 NCmemory date disp".

Note

(1) As for the device and directory, the setting values are retained even when the power is turned OFF. However, by selecting directory, when the number of characters in the directory path (full path) from the root directory exceeds 100, the directory path is not retained, but a previously set directory path containing 100 characters or less retained.
When the device is "Memory", the initial directory display is "Program".
When the device is "Ethernet", refer to "Initial directory when the Ethernet is selected" in "5.2 Selecting a Device, Directory and File" for details about the initial directory display.

Menus

Menus	Details
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.
Trnsfr 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.)
Trnsfr B->A	This copies the file in file setting column B (transfer origin) to the file setting column A (transfer destination). (The transfer origin file is not changed.)
Compare A:B	This compares the files in file setting column A and file setting column B.
Erase	This erases the selected file in file setting column.
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.
Dir create	This creates a new directory in the selected file setting column. When NC memory or NC memory 2 is selected as a device. this menu is grayed out and non-selectable.
Merge A->B	This adds the contents of the file in file setting column B to that of in file setting column A. (The file in file setting column B is not changed.)

Menus	Details
Close	This closes the pop-up window and quits this function.
MemCrd format	Format the memory card (front-side SD card).
DS format	Format the memory card DS (built-in SD card).
USB Mem format	Format the USB memory.
Stop	This interrupts the process (transfer, compare, etc.) during its execution.

Sub-menus of [File set] menu

Menus	Details
Memory	This selects NC memory.
Memory2	This selects NC memory 2.
HD	This selects the hard disk. This menu is not displayed on M800VS/M80V Series. (*1)
Serial	This selects the RS-232C device (PC, tape, etc.).
Memory card	This selects the front-side SD card.
DS	This selects the built-in SD card.
Ethrnet	This selects the Ethernet-connected host computer.
USB Memory	This selects USB memory.
Comment nondisp	This changes whether to show or hide the comment field. When the comment is hid, the menu is highlighted.
Program	This transfers a machining program as one file.
Program all	This transfers multiple programs as one file.
Multi-files	This menu is used when you select multiple files and perform the operation such as file transfer or file deletion.
Top jump	This moves the cursor to the top line.
Bottom jump	This moves the cursor to the last line.
List update	This updates the list contents. (The latest contents of the currently selected device and directory are listed.)

Menus	Details
Sort change	This changes the method that the list is sorted.
Return	This returns to the main menu.

(*1) When the HD mode operation on an industrial computer (IPC) is enabled with M80V Series, the menu [HD] becomes selectable and you can select files in the hard disk on the industrial computer. The availability of this function depends on the MTB specifications (parameter "#1760 cfgPR10/bit2").
Set the IP address of the industrial computer to the parameter #11005 to use this function.

Sub-menus of [Multi-files]

Menus	Details
Select range	This sets the last line of the multiple-file selection starting from the current cursor position.
Select all	This selects all files
Cancel all	This cancels the selection of all files.
Select	This selects the file where the cursor is currently located.
De-select	This cancels the file selection where the cursor is currently located.
fix	This fixes the file selection to return the main menu.
Return	This cancels the multiple-file selection to return the main menu.

Switching the valid area

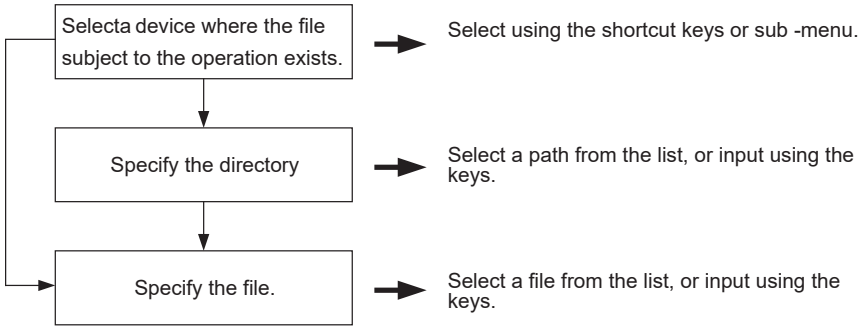
It is necessary to enable the areas including device, directory, and file name in the file setting column A or B in order to set those items.
Change the display area using the [←], [→], direction cursor keys or the [|←], [→|] tab keys.
For the touchscreen display, it is also possible to change the invalid area by touching the display area.

5.2 Selecting a Device, Directory and File

Edit – I/O

This section explains the file device, directory, and file name specification methods for the file transfer and erase commands etc. performed on the I/O screen of the edit (Edit) screen.

File operation procedure



Selecting methods for device, directory, and file name

Device	Designation target file	Designation method		
		Device	Directory	File name
NC memory NC memory 2	⬧Machining program ⬧User macro program ⬧Fixed cycle program	Select from the submenu	— (Default)	Key input in the input area, then press the [INPUT] key.
		Designate using the shortcut key.		Select from the list.
Devices other than above	All files	Select from the submenu	Key input in the input area, then press the [INPUT] key.	Key input in the input area, then press the [INPUT] key.
		Designate using the shortcut key.	Select from the list.	Select from the list.

Only machining programs can be stored on NC memory 2.
The device can be selected using the sub-menu or shortcut keys. (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 or NC memory 2) and file name.

⬧Designate the directory path or file name in the input area and press the [INPUT] key.
⬧Move the cursor in the list display section to the target directory or file name and press the [INPUT] key.

A wildcard (*) can be used for the file name.

Note

(1) If the number of full path characters exceeds 128 when the directory is being moved, it results in a movement failure.
(2) In the NC memory or NC memory 2, you cannot use the cursor movement key or [INPUT] key to move the directory.
(3) The file name and directory name must be designated using less than 64 characters. If the file name or directory name is designated using 64 or more characters, it may not be recognized as the file name or directory name.
(4) If a directory that contains a double-byte code file name such as a Japanese file name is displayed in list view, such a double-byte code file name is not displayed correctly. In addition, the number of file characters, comments, and number of stored characters is not displayed correctly.

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 parameter "#1166 fixpro". Furthermore, select "Memory" for the device, and "Program" for the directory.

Initial directory when the Ethernet is selected

If you reboot the system while the previously selected device is "Ethernet", or if you select Ethernet initially after startup, the initial directory will be the host directory that is set by the parameter (#9714, #9734, #9754 or #9774) when the HMI starts up.
The directory to be applied varies depending on the setting of the parameter "#9706 Host No.".

Operation method (Selecting an NC memory program)

(1) Press the [M] short cut key. on the I/O screen of the edit screen. → 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.

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.
Only when NC memory or NC memory 2 is selected as a device, 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 set.

[↑], [↓] [INPUT]

<When inputting the file name from the input area>

(3) Input the file name.

10013 [INPUT]

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.

■(Example 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".

■(Example 2) When programs 1.PRG to 1000.PRG, and 1 to 1000 exist in the USB memory.

Designate "*" in the file name: The target program is "1 to 1000".

Designate "*.*" in the file name: The target program is "1.PRG to 1000.PRG".

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.PRG, 10.PRG to 19.PRG, 100.PRG to 199.PRG, 1000.PRG".

Operation method (Selecting multiple files from the list)

You can select multiple files from the file list and perform the operation such as file transfer or file deletion. The status multiple files are being selected is canceled when a file transfer to the other device is performed or when any file among selected files is erased.

Note

(1) If you perform file operation such as copying or deletion to the file in the folder which multiple files are being selected, the file selection may be unintended condition due to change of the file list.
(2) When any of the menu operations ([Trnsfr A→B], [Trnsfr B→A], [Erase], Format menus, [Sort change], [List update]) is performed for the device which multiple files are being selected, the selection is canceled.
(3) Multiple-file selection is canceled when the I/O screen is closed.
(4) Directories can also be selected with the multiple-file selection function. However, directories can not be transferred, compared or erased.
(5) Multiple file selection is available in the range from the first file (including its directory) to 5000th file. When the menu [Select all] is pressed, the files up to the 5000th one are selected.

(1) Press the menu [File set].
→ The capacity and the input area of the active device are displayed.
The program list of the NC memory is displayed.
The menu [Memory] is highlighted.

(2) Press the menu "Multi-files" on the 2nd page.
→ The sub-menus of multiple-file selection function are displayed.

(3) Select files.

<When you select files one by one>

(a) Use the [↑], [↓], PAGE, or PAGE key to align the cursor with the target file, and press the menu [Select] or press [→] or [←] key.
→ The background color of the selected file changes.

(b) Repeat the operation (a) to select multiple files.

<When you select the files in a range>

(a) Use the [↑], [↓], PAGE, or PAGE key to align the cursor with the first file, and press the menu [Select range].
→ The menu [Select range] is highlighted.
The message "Move cursor to end posn and press INPUT." appears.

(b) Use the [↑], [↓], PAGE, or PAGE key to align the cursor with the last file.
→ When the cursor is moved, the background color of the file in the selection range changes.

(c) Press the [INPUT] key.
→ The range of the selected files is determined.
The background color for the range of the selected files changes.
The highlight of the menu [Select range] is canceled.

(4) Press the menu [fix] or [INPUT] key.
Multiple files selection has been determined.
The list display is canceled to back to the normal display, and the selected file name is displayed.

Canceling the file selection

(1) <When you cancel the selection for a single file>
Move the cursor to the file which you want to cancel the selection, and press the menu [De-select] or press [→] or [←] key.

<When you cancel the selection for all files>
Press the menu [Cancel all].

(2) Press the menu [fix] or [INPUT] key.

Operation method (Selecting a file from a device other than the NC memory or NC memory 2)

(1) Press a device selection shortcut key on the I/O screen of the edit screen.

(Example) [N]

→ The device name changes to "USB memory".
The root directory (/) appears in the directory.

(2) Press the menu [File set].

By pressing the operation menu at this point, it is possible to change the device.
Only when the device functions as the memory, the menu [Program] or [Program all] is valid.

→ The program list of the USB memory and the input area appear.

<When selecting the directory from the list>

(3) Move the cursor to the directory to be selected and set.

[↑], [↓] [INPUT]

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 the directory from the input area>

(3) Input the directory path.

(Example) /PRG/PRECUT [INPUT]

→ 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 set.

[↑], [↓] [INPUT]

<When inputting the file name from the input area>

(4) Input the file name.

10013.PRG [INPUT]

5.3 Transferring a File

Edit – 

This section explains the method used to transfer files between different devices, or between the same device on the I/O screen of the edit (Edit) screen.
Refer to "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 on the I/O screen of the edit screen.

(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.

(5) Press the menu [Trnsfr A->B].
Press the menu [Trnsfr B->A] when transferring from device B to device A.

(6) Press the [Y] or [INPUT] key. → File transfer is commenced. The process progress display area displays message indicating that transfer is in progress, and also indicates the transfer direction.
The data being merged appears in the input/comparison data display area 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 "%". "%" and "EOR" are represented by "%" in ISO. The programs created on the edit screen are stored in the NC memory or NC memory 2 in a "CR, LF" format, however, the programs created with external devices may be stored in an "LF" format. The actual codes for EIA are "EOB (End of Block)" and "EOR (End of Record)".

❗ To prevent influence from data omission and data transformation in the communication circuit, always verify the data after inputting and outputting machining programs.

Precautions

■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) When files are input to the NC memory or NC memory 2, or files in the said device are compared, if the file format size on the NC memory/NC momory 2 differs from the other side (i.e. the maximum number of registrations differs between the NC memory/NC memory 2 and the other side), processing is carried out matched to the smaller one.

(Example 1) If 200 files attempt to be input to an NC of 1000-file format, 200 files are registered.

(Example 2) If 1000 files attempt to be input to an NC of 200-file format, the files up to the 200th can be registered and an error message appears. (The remaining files are not registered.)

(4) Do not perform any file operation on the transfer source and destination files during the file transfer. The file may be damaged with the file operation.

■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 should be 250 characters or less.

(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
Up to 32 characters are allowed for the file name to create in or transfer to NC memory.

(5) If the file whose name includes lower case alphabet is transferred to NC memory, the lower case letter will be converted into the capital letter.

(6) When using tape, carry out parity V adjustment to improve the reliability of the tape format. Then use with the parameter "Parity V" validated.

(7) When the MTB 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

(8) Transferring or verifying the multiple files between the external device connected serially and that other than the serial connection.

(9) With machining program created by the MELDAS500 Series or earlier model, "EOB" is registered as "LF". However, when these programs are stored in the M800/M80 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 M800/M80 Series NC memory, the memory capacity may be exceeded.

(10) 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).

(11) If the file to be transferred (input) is being edited on the custom screen, the operation message "Can't write into being edited file" appears and may not be transferred (input).

(12) When "#8936 Delete leading 0" is set to "1", if a file name consisting only of numerical figures is input, "0" at the head of the file name is deleted to be checked.
However, if the file name is out of the range between 1 and 99999999, the leading "0" is not deleted. This also applies to MTB macros.
Therefore, for example, while the program "1" is in automatic operation, if a file name to transfer (input) is named "0001", the operation message "Executing automatic operation" appears, and transfer cannot be carried out.
In the case of fixed cycles, regardless of the setting of "#8936 Delete leading 0", the name of the file to transfer must be 9 digits, and it must not start from "0". Attempting to transfer a file whose name does not satisfy the conditions above will cause an error.

(13) If the file to be transferred (input) is targeted for edit lock (edit lock B: 8000 to 9999 or edit lock C: 9000 to 9999), the operation message "Edit lock B" or "Edit lock C" is displayed, causing a transfer failure.
If "#8936 Delete leading 0" is set to "1" and the program name consisting only of numerical figures is input, "0" at the head will be deleted and checking will be performed. (Example: If the transferring (inputting) file name is "008000", the operation message "Edit lock B" appears and it cannot be transferred.)

(14) When the multi-part system program management is valid (#1285 ext21/bit0 =1) and "#1285 ext21/bit2" (Multi-part system program generation and operation) is set to "0", the machining program's transfer to NC memory or NC memory 2 from the other device is executed across all part systems in batch. When the parameter is set to "1", only displayed part system's machining program will be transferred.

(15) When the multi-part system program management is valid (#1285 ext21/bit0 =1) and "#1286 ext22/bit5" (Selection of multi-part system program input/output method) is set to "0", the machining program's transfer from NC memory or NC memory 2 to the other device is executed across all part systems in batch. When the parameter is set to "1", only displayed part system's machining program will be transferred.

(16) The transfer range from the external device other than serial to NC memory or NC memory 2 will change depending on the value set in the parameter "#19006 EOR Disable".

<When the parameter "#19006 EOR disable" is set to "0">

The head line will not be transferred.

Up to "%" in the second line onward will be transferred, and data after "%" will not be transferred.

If there is no "%", all data will be transferred.

<When the parameter "#19006 EOR disable" is set to "1">

All file contents will be transferred.

(17) When multiple files are designated to be transferred in the following ways, the transfer is executed excluding the programs which are subject to the edit lock in accordance with the status of Edit lock B and C.

- A wildcard "*" is used in the file name.
- Multiple files are selected from the list.

(18) If the devices and the directories listed below are designated for the destination to transfer a file, a temporary file is created.
When the transfer is successfully completed, the name of the temporary file is changed to the designated one. (*1)
When the transfer is performed to overwrite an existing file, the temporary file is replaced with the existing one.

(1) Transfer destination device: Memory
Directory: Machining program

(2) Transfer destination device: DS, memory card (M800VS/M80V Series), USB memory (M800VS/M80V Series)

(*1) A temporary file remains in the following cases:

- When the transfer has terminated abnormally, for example, because the destination's capacity is insufficient.
- When the transfer has been interrupted.

The name of a temporary file starts with the symbol "~", and it consists of 11-digit numbers combined. The file can be deleted in the same manner as a normal file.

5.4 Comparing Files (Compare)

Edit – 

This section explains method used compare files after transferring on the I/O screen of the edit (Edit) screen.
Refer to "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 on the I/O screen of the edit screen.

(2) Designate the device, directory and file name to be compared.

(3) Use the [→] or [→|] key to select file setting column B.

(4) Designate the other device, directory and file name to be compared.

(5) Press the menu [Compare A:B]. → 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.

Note

(1) Files that can be compared are text files only. Correct outcome will not be obtained through binary file comparison.

(2) In a file comparison, even if "#8936 Delete leading 0" is set to "1", when comparing one separate file that contains one program, "0" at the head of the program name is not deleted. Designating the file name is required.
However, in the case of serial/batch file, comparison is carried out with a leading zero in the program name deleted.

(3) Designating "*" for the compared file name causes "Setting error" when the comparison is executed with serial. Designate no file name or designate a target file name.

(4) When multiple files are designated for the transfer source file in the following ways, the comparison is executed excluding the programs which are subject to the edit lock in accordance with the status of Edit lock B and C.

- A wildcard "*" is used in the file name.
- Multiple files are selected from the list.

5.5 Erasing a File

Edit – 

This section explains the method used to erase files on the I/O screen of the edit (Edit) screen.
Refer to "5.2 Selecting a Device, Directory and File" for details of the device, directory, and file specification method.

Operation method (Erasing a file in file setting column A)

(1) Use the [←] or [|←] key to select file setting column A on the I/O screen of the edit screen.

(2) Designate the device, directory and file name to be erased.

(3) Press the menu [Erase].

(4) Press the [Y] or [INPUT] key.

Note

(1) If the file to be erased is running, an error will occur, and the file will not be erased.
(2) If the file to be erased is being edited on the custom screen, the operation message "Can't erase designated file" appears and may not be erased.
(3) When multiple files are designated to be erased in the following ways, the erasing is executed excluding the programs which are subject to the edit lock in accordance with the status of Edit lock B and C.
- A wildcard "*" is used in the file name.
- Multiple files are selected from the list.

(4) The image data associated with the file is also erased. However, when both of the parameters "#1285 ext21/bit0" (Multi-part system program management) and "#1285 ext21/bit2" (Multi-part system program generation and operation) are set to "1", and when a file of the same name exists in the part system other than selected one, the image data associated with the file will not be erased.

Operation method (Erasing 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 "Erasing a file in file setting column A".

5.6 Changing a File Name (Rename)

Edit – 

This section explains the method used to change the file name on the I/O screen of the edit (Edit) screen.
Refer to "5.2 Selecting a Device, Directory and File" for details of the device, directory, and file specification method.

Changing a file name

(1) Use the [←] or [|←] key to select file setting column A on the I/O screen of the edit screen.

(2) Designate the original device, directory and file name.

(3) Use the [→] or [→|] key to select file setting column B.

(4) Designate the new device, directory and file name.

(5) Press the menu [Rename A->B].

(6) Press the [Y] or [INPUT] key.

Note

(1) Make sure the original and new devices are the same.

(2) If the file to be renamed is running or program restarting, an error will occur, and the name will not be changed.

(3) If the file to be renamed is being edited on the custom screen, the operation message "Can't rename designated file" appears and may not be changed.

(4) 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.

(5) The directory where the file in operation search/check search is stored cannot be renamed. The operation message "Can't rename designated file" appears.

(6) The image data associated with the file is also renamed. However, both of the parameters "#1285 ext21/bit0" (Multi-part system program management) and "#1285 ext21/bit2" (Multi-part system program generation and operation) are set to "1", and when a file of the same name exists in the part system other than selected one, the image data associated with the file will not be renamed.

5.7 Creating a Directory

Edit – 

This section explains the method used to newly create a directory.
Refer to "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.

(3) Select a directory from the list.

(4) Set the new directory in the file setting column A. → The directory contents appear.

(5) Press the menu [Dir create].
The directory can also be created in file setting column B. → A new directory is created in the directory designated in step (3).

Note

(1) Make sure that the directory path is less than 64 characters. If the directory name is designated using 64 or more characters, it may not be recognized as the directory name.

5.8 Other Functions

Edit – 

5.8.1 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 on the I/O screen of the edit (Edit) screen.
Refer to "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 on the I/O screen of the edit screen.

(2) Designate the merge destination device, directory, and file name.

(3) Use the [→] or [→|] key to select file setting column B.

(4) Designate the merge source device, directory, and file name.

(5) Press the menu [Merge B->A].

(6) Press the [Y] or [INPUT] 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.;
(FILE B)
G91 G28 XYZ
F1000;
G01 X200. Y200.;
M02;
%
```

Details of file in file setting column A before merge.

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.
(2) If the merge destination file (file in file setting column A) is being edited on the custom screen, the operation message "Can't write into being edited file" appears and may not be changed.
(3) The devices that can be merged are the memory, HD (*1), memory card, DS and USB memory.
(4) Do not perform the file operation on the merge source and destination files during the merge. The file may be damaged with the file operation.

(*1) Hard disk which is displayed by connecting with an industrial computer is excluded.

5.8.2 List of File Names

There is a directory for each type of data in NC memory and NC memory 2.

Each directory and file name (fixed) in the device is shown below.

NC memory 2 has the only directory for machining programs.

Do not change the extensions (.XXX) when storing in a device other than NC memory or NC memory 2.

Data type	NC memory directory path	File name
Machining program	/PRG/USER /PRG2/USER	(Program No.)
Fixed cycle program	/PRG/FIX	(Program No.)

5.8.3 Edit Lock B and C

This function prohibits editing, erasing, etc., of the machining programs B and C, and protects the machining programs in the device [Memory].

Machining program		Edit Lock B	Edit Lock C
A	1 to 7999 10000 to 99999999	---	---
B	(user standard program) 8000 to 8999	Editing prohibited	---
C	(MTB custom) 9000 to 9999	Editing prohibited	Editing prohibited

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	○	×	×	○	○	×
Monitor	Buffer Correction	○	×	×	○	○	×

Note

(1) When the multiple part system specification is valid, the machining programs of all the part systems are protected by edit lock B and C.

(2) When multiple serial files or a wildcard (*) is designated for input/output function, programs except subject to edit lock will be processed.

(3) When a multiple programs are input to the NC memory or NC memory 2 without designating the file name like serial input or program batch input, the message "Edit lock B" or "Edit lock C" appears and the input is interrupted if the program which is subject to edit lock is detected.

(4) A machining program in the USB memory is assumed as machining program A whatever its file name is, it is not subject to the protection of Edit Lock B or C.

5.8.4 Program Display Lock

Display and operation search of the program stored NC memory or NC memory 2 on the monitor or edit screen are affected as follows according to the setting of the parameter "#1122 pglk_c" (Program display lock).

Influences of program display lock parameter on monitor screen

Screen operation #1122 pglk_c	1 (Display: Prohibited, Search: Possible)	2 (Display: Prohibited, Search: Prohibited)
Buffer display of operating program	The program contents are not displayed. Only "%" is displayed.	
Display when [Program display] highlights on the 2D check full-screen	The program contents are not displayed. Only "%" is displayed. (Drawing is performed.)	
Display when [Program display] highlights on Trace full-screen		
When the menu [Edit] was pressed	The operation message "Program display lock C" appears. The window is opened. (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		
Search from the operation search window	The search can be executed.	The search cannot be executed. The operation message "Program display lock C" appears. The window is not closed. The contents of input area are not cleared.
Search from the restart search window		
Search from the top search window (of restart search)		

Influences of program display lock parameter on edit screen

Screen operation #1122 pglk_c	1 (Display: Prohibited, Search: Possible)	2 (Display: Prohibited, Search: Prohibited)
The file is automatically opened when changing the edit screen (*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.)	
Display when [Program display] highlights on the 2D check full-screen	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. The contents of input area are not cleared.

(*1) When changing the edit screen, the file opened immediately before or operation searched/check searched is automatically opened.

5.8.5 Data Protection Keys

Data protection key prevents various data from being set or erased.
The key name differs according to the MTB. Refer to the instruction manual issued by the MTB for details.

The following data can be protected with the data protection key:

- All tool data and the coordinate system preset data using the origin set
- User parameters and common variables
- Machining programs
- Files in the memory card
- Files in the DS

The data protection key is enabled to prohibit the setting, erasing, etc. while the PLC signal is OFF (B contact).
Data protection key has higher protection level than that of Data protection by user's level.

5.8.6 Leading Zero

When creating a new machining program

When the parameter "#8936 Delete leading 0" is set to "1", if a file name consisting only of numerical figures is input, the file is created with "0"at the head of the file name deleted.
However, if the file name is out of the range between 1 and 99999999, the leading "0" is not deleted. This also applies to MTB macros.
In the case of fixed cycles, regardless of the setting of "#8936 Delete leading 0", a file name to create must be 9 digits, and it must not start from "0". Attempting to create a name not following the conditions above will cause an error.
When "#8936 Delete leading 0" is set to "1", if a file name consisting only of numerical figures is input, the leading "0" is deleted when checking whether the file exists or not, and when checking edit lock B/edit lock C.
(Example) For a directory that has a file "1", if a file name "0001" is attempted to be input as a new file, an error will occur.

Example of deleting of leading zero

Input file name	Created file name	Remarks
0012345678	12345678	The leading "0"s are deleted
0123456789	0123456789	As this name is out of the rage between 1 and 99999999, the leading "0" is not deleted
0123A	0123A	As a nonnumerical character (alphabet) is included, the leading "0" is not deleted
0123.PRG	0123.PRG	As a nonnumerical character (extension) is included, the leading "0" is not deleted

When transferring, or comparing files

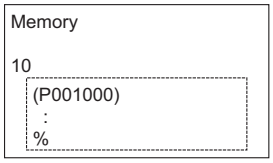
In a file transfer with "#8936 Delete leading 0" set to "1", only when the file name of the transfer destination consists only of numerical figures, the leading "0" is deleted when the file is transferred.
However, if the file name is out of the range between 1 and 99999999, the leading "0" is not deleted. This also applies to MTB macros.
In the case of fixed cycles, regardless of the setting of "#8936 Delete leading 0", the name of the file to transfer must be 9 digits, and it must not start from "0". Attempting to transfer a file whose name does not satisfy the conditions above will cause an error.
In a file comparison, even if "#8936 Delete leading 0" is set to "1", when comparing one separate file that contains one program, "0" at the head of the program name is not deleted. Designating the file name is required.
However, in the case of serial/batch file, comparison is carried out with a leading zero in the program name deleted.

■(Example 1) One file has one program

(1) The files are transferred from the USB memory to NC memory.

A device: USB memory Directory: / File name: 001000	⇨	B device: Memory Directory: Program File name: 000010

The files are transferred to the NC memory as follows.
The leading zero is deleted from the file name.

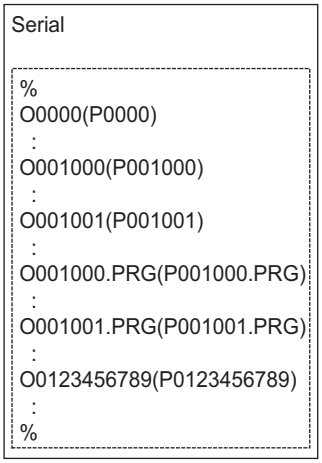


(2) In order to carry out comparison, change the file name of the transfer destination to "10".
(In a comparison of one separate file that has one program, as "0" at the head of the program name is not deleted, if the file name remains "000010", it will be compared with the file "000010" of the transfer destination.)

A device: USB memory Directory: / File name: 001000	⇔	B device: Memory Directory: Program File name: 10

■(Example 2) Serial/Batch file

(1) The serial of the transfer destination has files as below.

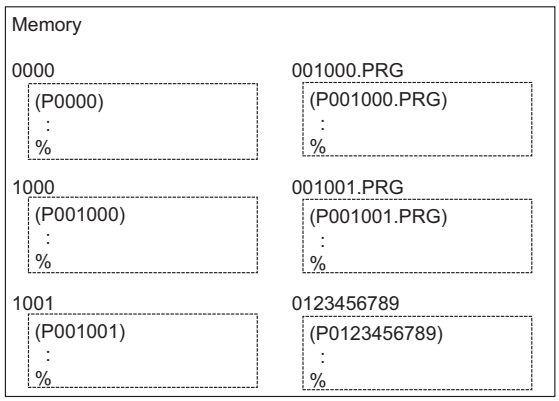


(2) The files are transferred from the serial to NC memory.

A device : Serial	⇨	B device: Memory
Directory: /		Directory: Program
File name: (blank)		File name: (blank)

The files are transferred to the NC memory as follows.

- Files whose leading zeros are deleted: 001000, 001001
- Files whose leading zeros are not deleted as the names are out of the range between 1 and 99999999: 0000, 0123456789
- Files whose leading zeros are not deleted as the names contain nonnumerical characters: 001000.PRG, 001001.PRG



(3) Comparison is carried out. (As the serial/batch files are compared with the leading zeros deleted, changing file names is not required.)

A device : Serial	⇔	B device: Memory
Directory: /		Directory: Program
File name: (blank)		File name: (blank)

5.9 The Batch Input/Output the Machining Program of NC Memory/NC Memory 2

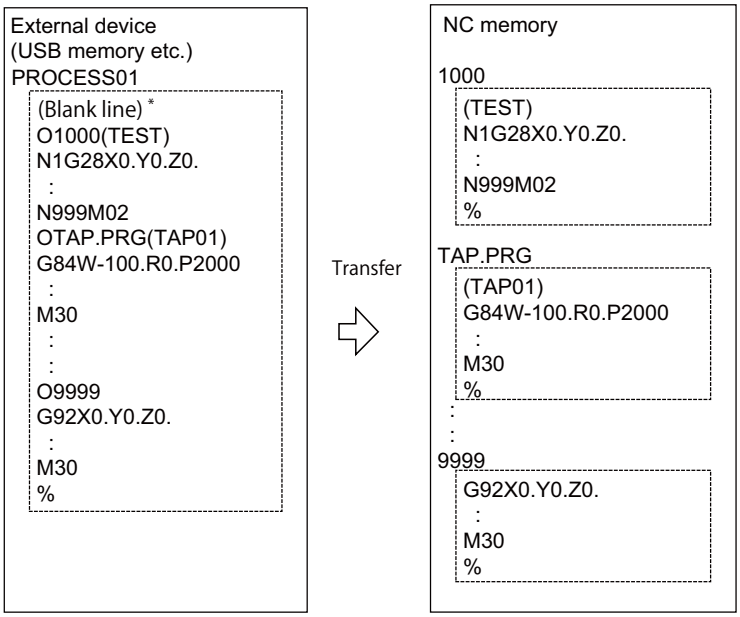
Edit – 

On the I/O screen of the edit (Edit) screen, one file which consists of two or more machining programs can be transferred to NC memory or NC memory 2 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.

The target external devices are HD, memory card, DS, Ethernet, serial and USB memory.

Batch input and verify to NC memory/NC memory 2

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).
Comment is not added after a program name when a program is output to an external device, however, if there is comment after a program name when a program is input, the comment is input to the head of the first part system.

(1) Use the [←] or [|←] key to select file setting column A on the I/O screen of the edit screen.

(2) Designate the transfer source device and directory.

(3) Designate the file which consists of two or more machining programs.

(4) Use the [→] or [→|] key to select file setting column B.

(5) Designate the transfer direction device [Memory].

(6) Press the menu [Program all].
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 [Trnsfr A->B].

(8) Press the [Y] or [INPUT] key.
When the same machining program already exists in transfer direction device, the operation message "Overwrite this file?(Y/N)" appears. The transferring the following program is continued by selecting [Y] or [N] key.

➡ File transfer is commenced. Transmitting data is displayed in input data display column. The operation message "Transfer complete" appears upon the completion of transmission.

(9) Press the menu [Compare A:B].

➡ The file comparison starts. The verifying data is displayed. A message appears when the comparison is completed. When the verification error occurs, the error block is displayed in verification data display column and the operation message "Compare error. Compare next file?(Y/N)" appears.

Note

(1) When programs are input in a batch to NC memory or NC memory 2, always select [Program all] for NC memory or NC memory 2 as transfer destination. If [Program all] is not selected, one file is transferred (No batch).

(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 "5.8.3 Edit Lock B and C".)

(3) When the protection of the machining program is valid, the program cannot be transferred/verified. (Refer to "5.8.5 Data Protection Keys".)

(4) The program during automatic operation, program restart or program checking cannot be overwritten. The operation message "Executing automatic operation", "Program restarting" or "Program checking" appears, and transfer is interrupted.

(5) If the transfer destination includes the file being edited on the custom screen, the operation message "Can't write into being edited file" appears and the transfer may be interrupted.

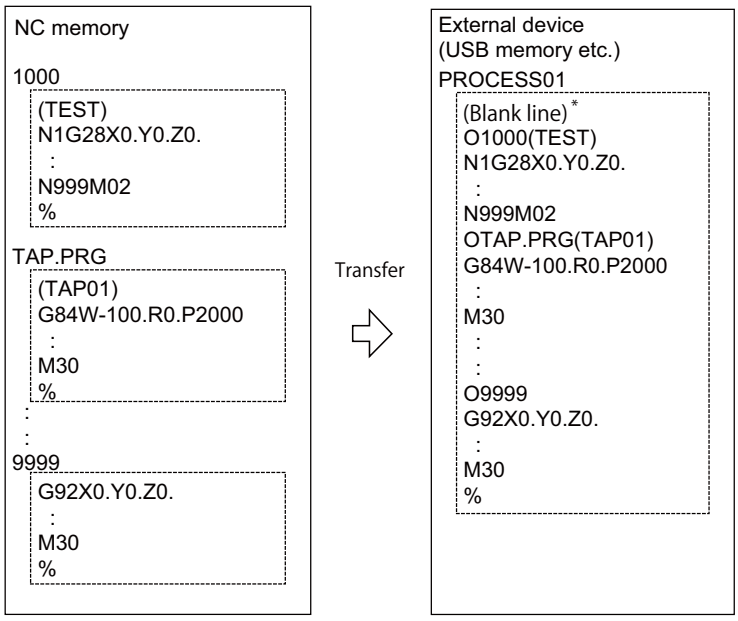
(6) The first line of the transfer source file is ignored.

(7) 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 all	Program
File name	(Null or file designation)	(Null or file designation)

Batch output and verify from NC memory/NC memory 2

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 on the I/O screen of the edit screen.

(2) Designate the transfer source device [Memory].

(3) Press the menu [Program all].
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.

(6) Designate the transfer direction file to input multiple machining programs.
When the file name is not designated, output to the file "ALL.PRG" of the designated directory.

(7) Press the menu [Trnsfr A->B].

(8) Press the [Y] or [INPUT] key.
When the same machining program already exists in transfer direction device, the operation message "Overwrite this file?(Y/N)" appears. The transferring the following program is continued by selecting [Y] or [N] key.

➡ File transfer is commenced. Transmitting data is displayed in input data display column.
The operation message "Transfer complete" appears upon the completion of transmission.

(9) Press the menu [Compare A:B]. → The file comparison starts. The verifying data is displayed. A message appears when the comparison is completed. When the verification error occurs, the error block is displayed in verification data display column and the operation message "Compare error. Compare next file?(Y/N)" appears.

Note

(1) When programs are output in a batch from NC memory or NC memory 2, always select [Program all] for NC memory or NC memory 2 as transfer source. If [Program all] is not selected, the program is transferred by one program per one file.

(2) When the programs which are subject to edit lock exists in NC memory or NC memory 2, those files are not transferred. (The files are transferred excluding those files.) The verification is executed excluding the programs which are subject to edit lock. (Refer to "5.8.3 Edit Lock B and C".)

(3) When the protection of the machining program is valid, the program cannot be transferred/verified. (Refer to "5.8.5 Data Protection Keys".)

(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 all	Program
File name	(Null)	* (Wildcard)

Precautions

(1) When the machining program batch input/output function of NC memory or NC memory 2 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 [Program all] is selected for NC memory or NC memory 2, it is not possible to delete, rename and merge files. When the files are deleted in a batch, select [Program] for directory, and a wildcard (*) for file name.

(4) When you use the all-program batch input/output function for the NC memory or NC memory 2, the programs except for those protected are output. If the NC memory contains any protected program, the operation message "Transfer complete. Some file not output" appears.

(5) For other precautions, refer to "5.3 Transferring a File".

5.10 Formatting Memory Card/DS/USB Memory

Edit – 

This section explains how to format the memory card (front-side SD card), DS (built-in SD card) and USB memory.

(Example) Formatting memory card

(1) Press the menu [MemCrd format]. → The message confirming the format "OK?(Y/N)" appears.

(2) Press the [Y] or [INPUT] key. → The memory card is formatted.
During format, the message "Executing format" appears and the message "Format complete" appears when the format is completed.

When formatting DS or USB memory, press the menu [DS format] or [USB Memformat] in the procedure (1).

Note

(1) When the capacity of memory card, DS and USB card are less than 2GB, the format is executed with FAT16, and when their capacity are 2GB to 32GB, format is executed with FAT32.
When formatting the memory card, DS or USB memory whose capacity exceeds 32GB, the message "Format error" appears and the format cannot be executed.

(2) The format cannot be executed during automatic operation.

6

Setting a Tool and a Workpiece

This chapter explains the details of tools and workpieces based on respective menus on the setup (Setup) screen.

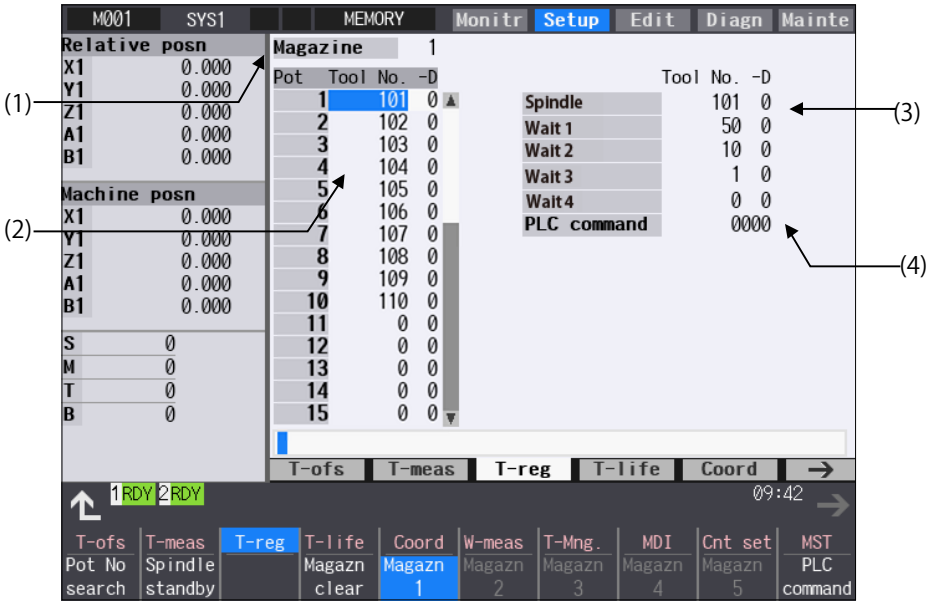
6.1 Setting a Tool

Refer to the instruction manual issued by the MTB for details on the tool setting.

6.2 Registering a Tool (Tool Registration Screen)

Setup – T-reg

A tool No. is assigned to each tool to make the tools installed on the machine recognizable to the NC on the tool registration (T-reg) screen of the Setup screen. 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 200 tools can be registered for each magazines. When making an arbitrary setting, there are a maximum of 5 magazines, and up to 600 tools can be registered for all magazines.
This function differs depending on the specifications of the MTB. Refer to the instruction manual issued by the MTB for details.



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 items		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 manual issued by the MTB for details on the "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	This displays spindle or standby tool No. Screen display varies depending on the MTB specifications.
(4)	PLC command	This command is used to input data and perform sequence processing using the PLC program.

Menus

Menus	Details
Pot No search	Magazines are displayed sequentially with the selected pot number on the top of them. The cursor moves to the data of the configured pot number, and tool number can be entered.
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.
Magazn clear	This erases all the currently selected magazine tool data (tool number and D). <Note> •Unselected magazine tool data is not erased. •The spindle and standby data cannot be erased.
Magazn 1 : Magazn 5	This designates the magazine No. to be displayed on the screen. The displayed menu differs according to the machine specifications. For example, when there are two magazines, only the menus [Magazine 1] and [Magazine 2] display.
PLC command	The cursor moves to the PLC command setting area and the mode changes to the setting mode.

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-dup check".

<Priority of duplication check>

Operation	Setting of the operation parameter "#8922 T-reg-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 "6.2.1 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 "6.2.3 Setting and 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

When the value of the parameter "#8922 T-reg-dup check" is out of the range, the check of setting "0" is performed (Executing the duplication check for all valid magazines).

6.2.1 Registering a Tool in the Magazine Pot

Operation method (Selecting a magazine No.)

(1) Press the menu [Magazn 2]. → The tool data of the set magazine No. appears.

Note

(1) The No. of magazines differs according to the machine specification.

Operation method (Registering the tool in the magazine pot)

(1) Designate the pot No.
[Pot No search] 21 [INPUT]

(2) Input the tool number.
50 [INPUT]
When the [INPUT] 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 invalid magazine, spindle/standby.

Note

(1) Move the cursor with [→|] key to set the "D" data. Refer to the manual issued by the MTB for details on the "D" functions and purposes.
(2) The duplication check is executed for only the tool No. The check does not depend on the setting of "D".
(3) If parameter "#8922 T-reg-dup check" is set to "1", the operation message will not appear even when the tool No. duplicates.
(4) The magazine to be checked varies depending on the setting of the parameter "#8922 T-reg-dup check".

6.2.2 Setting the PLC Command

Operation method (Setting the PLC command)

(1) Set the value after the menu [PLC command].
[PLC command] 12 [INPUT]
→ 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 MTB specifications.

(2) Press either the menu [PLC command], [◁] key or [↵] key to complete the setting.
→ The PLC command setting mode is released, and the menu highlight returns to normal.

Note

(1) Select the menu [PLC Command] again before pressing the [INPUT] key, then the menu highlight returns to normal, and the PLC command setting mode becomes disable.

6.2.3 Setting and Erasing the Tool No. of Spindle/Standby Tools

Operation method (Setting the tool No. of spindle/standby tools)

(1) Press the menu [Spindle standby].

(2) Move the cursor to the data to be set using the [↑], [↓] keys.

(3) Input a new tool No.
50 [INPUT]

(4) Press either the menu [Spindle standby], ◁ key or ➡ key to complete the setting. → The spindle/standby setting mode is canceled.

Note

(1) When the menu [Spindle standby], ◁ key or ➡ key is pressed before pressing the [INPUT] key, the spindle/standby setting mode will be canceled and the set data will be ignored.

(2) Move the cursor with →| key to set the "D" data.

Operation method (Erasing the tool No. of spindle/standby tools)

(1) Press the menu [Spindle standby].

(2) Move the cursor to the tool No. to be erased and input "0".
0 [INPUT]

Note

(1) Move the cursor with →| key and set "0" to erase the "D" data as the same manner as the tool No.

6.2.4 Erasing the Tool Registration Data

Operation method (Selecting a magazine number and erasing the tool registration data.)

(1) Press the menu [Magazn 2].

(2) Press the menu [Magazn clear].

(3) Press the [Y] or [INPUT] key. → All selected magazine tool data (tool number and D) is erased.
Spindle/standby tool data is not erased.

6.3 Measuring a Tool (Tool Measurement Screen)

Setup – 

(M system)

Setup – 

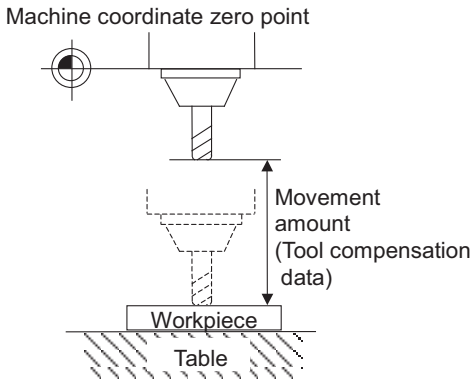
(L system)

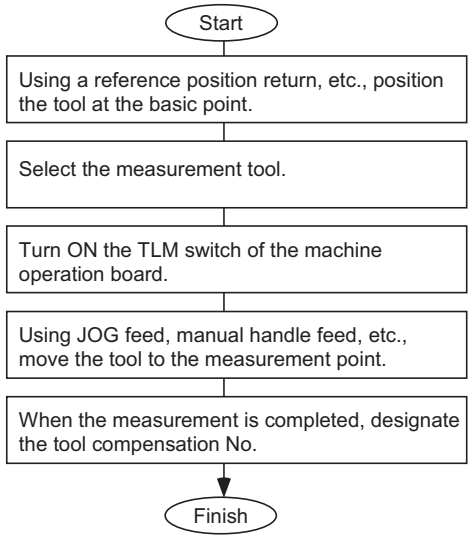
6.3.1 Tool Measurement (M System)

On the tool registration (T-reg) screen of the setup (Setup) screen, the movement distance from the basic point to the measurement point is measured by moving the tool to the measurement point manually, and this value can be set as the tool compensation amount.
Manual numerical value command can be executed when manual numerical value command window is displayed by inputting an address key such as MST, etc.
There are two types of tool measurement: Manual tool length measurement 1 and manual tool length measurement 2.

■Manual tool length measurement 1

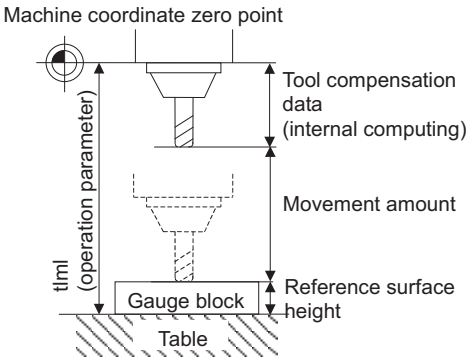
When the tool is at the machine coordinate zero point, the distance from the tool tip to the measurement point (workpiece upper end) is measured, and can be set as tool compensation data.

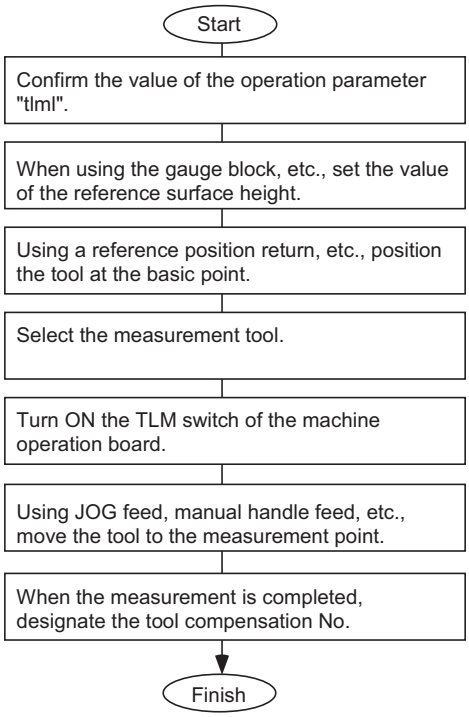




■Manual tool length measurement 2

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.

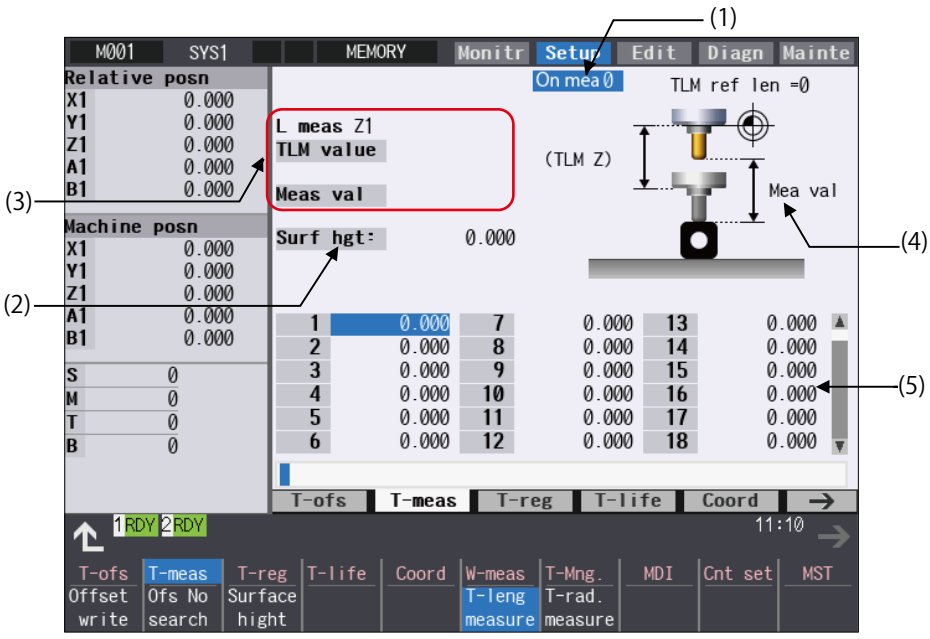




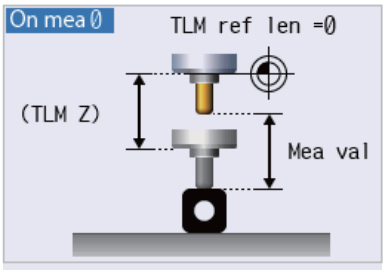
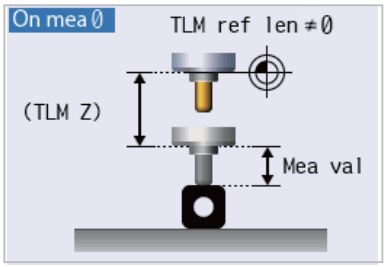
Note

(1) The tool measurement type is determined based on TLM reference length (parameter "#2016 tlml+" (TLM standard length)) of measurement axes (tool length measurement axis and tool radius measurement axis) at each measurement mode. When parameter "#2016 tlml+" (TLM standard length) is set to "0", and "Surface hight #0 = 0" is set, the tool measurement type is manual tool length measurement 1.

■Screen image



Display items

	Display items	Details
(1)	Manual measurement status display	This displays the manual measurement status. Refer to "Manual measurement status display" described later in this section.
(2)	Reference surface height	This displays a value for the reference surface height. Setting range: -99999.999 to 99999.999 (mm)
(3)	Counter display	TLM value: This displays the value during measurement. (*1) 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 manual tool length measurement 1: TLM value - Reference surface height For manual tool length measurement 2: TLM value - Reference surface height + TLM reference length For tool radius measurement, measurement result is displayed in absolute position.
	L meas: (Axis name) R meas: (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.
(4)	Guide graphic	Displays the measurement image. The contents of the guide drawing will differ depending on the manual tool length measurement (1 or 2). Manual tool length measurement 1 guide drawing  Manual tool length measurement 2 guide drawing 
(5)	Tool compensation data	Displays the tool compensation data. The contents of the tool compensation data to be displayed vary depending on the tool compensation type (I, II, or III). When tool compensation type III is selected, the display is the same as for tool compensation type II. Cursor movement, data input and part system changing operation are same as tool compensation types I and II.

(*1) 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.

Note

(1) If the parameter "#11017 T-ofs set at run" is set to "1", the tool compensation amount data can be measured in the automatic operation mode or operation pause mode.

Menus

Menus	Details
Offset write	This writes the value displayed in "Mea value" as the tool compensation amount. The wear amount is cleared to "0" for the tool compensation type II. The tool compensation amount cannot be written in when the menu [Ofs No search] or [Surface hight] is highlighted. <Note> ♦When the parameter "#8924 Meas. confirm msg" is "1", this menu is highlighted and the operation 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 highlighting is canceled and the operation message is erased.
Ofs No search	When the compensation No. is set and the [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.
Surface hight	This sets the reference surface height data. (The cursor moves to the reference surface height.) When the data is set in the input area and the [INPUT] key is pressed, the reference surface height is set.
T-leng measure	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 or III.)
T-rad. measure	This changes the mode to the tool radius measurement mode. (The tool measurement type is valid only for manual tool length measurement 2.) (The cursor moves to the radius dimension position for tool compensation type II or III.)

Carrying out tool length measurement

(1) Turn ON the "Tool length measurement 1" signal (YC20). → "On mea 0" is displayed on the tool measurement screen of the setup screen.

(2) Press the menu [T-leng measure]. → The menu is highlighted, and the tool length measurement starts. The measurement values are displayed in "Meas value" during the measurement.
The menu [T-leng measure] 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.
(Example) [Ofs No search] 12 [INPUT]
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 [Offset write]. → The measurement values are written as a tool compensation amount.
The wear amount is cleared to "0" for the tool compensation type II.

(6) Turn OFF the "Tool length measurement 1" signal (YC20) to finish measuring.

Note

(1) The measurement values are not written if the cursor is in a position other than "Length" or "Radius".

(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.

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 [T-rad. measure] instead of [T-leng measure].)

6.3.2 Tool Measurement (L System)

Manual tool length measurement 1 or 2 can be performed on the tool registration (T-reg) screen of the setup (Setup) screen. Either one is selected depending on the setting of measurement switch and operation mode.

Manual numerical value command can be executed when manual numerical value command window is displayed by inputting an address key such as MST, etc.
There are two types of the manual tool length measurement 1: normal operation mode and simple operation mode, which can be selected by the parameter "#8957 T meas (L)-Simple".

#8957	Manual tool length measurement 1 operation mode
0	Normal operation mode
1	Simple operation mode

■Manual tool length measurement 1 (when measurement switch is ON)

There are three methods for the manual tool length measurement 1, and the values to set for the measurement zero point will differ. Set the measurement method by the combination of the parameters "#11087 Meas basic point" and "#1284 ext20/bit2".

#11087 Meas basic point	Method	#1284 ext20/ bit2	Measurement zero point
0	Machine zero point designation method	-	Machine coordinate zero point
1	Parameter designation method	-	The coordinates set by "#2015 tlml-"
2	Workpiece coordinate system offset designation method	0	Position shifted by the amount of workpiece coordinate system offset (modal) from the machine coordinate zero point
		1	Position shifted by the total amount of the following offsets from the machine coordinate zero point •Workpiece coordinate system offset (modal) •External workpiece coordinate offset •Workpiece coordinate system shift •Workpiece base point offset (*1)

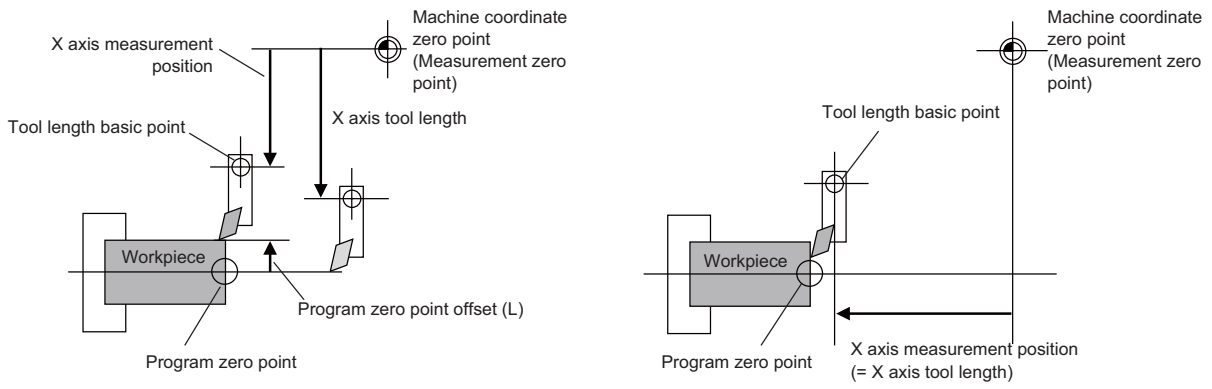
(*1) For details on the workpiece base point offset, refer to "6.9 Setting Workpiece Coordinate System Shift (L System)".

[Machine zero point designation method (#11087 = 0)]

In this method, the machine coordinate system zero point is the measurement point.

Move the tool to the face or side, and calculate the tool compensation amount from the measurement position (machine position) and program zero point offset (L).

Set the program zero point offset (L) by pressing the menu [Zero-pt offset] on the [Setup] - [T-meas] screen.



The tool length is automatically calculated by the following formula:
Tool length = Measurement position - Program zero point offset (L)

Note

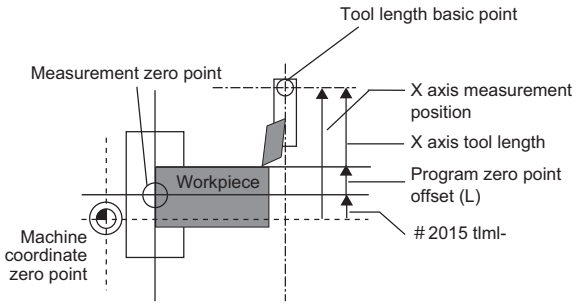
(1) For the program zero point offset (L), set the diameter value for a diameter axis (#1019 dia = 1) or the radius value for a radius axis (#1019 dia = 0).

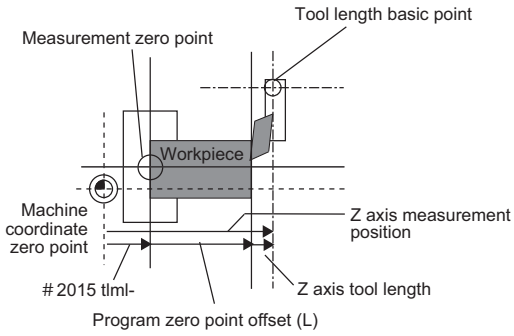
[Parameter designation method (#11087 = 1)]

In this method, the offset from the machine coordinate system zero point set in "#2015 tlml-" is treated as the measurement zero point.

Move the tool to the face or side, and calculate the tool compensation amount from the measurement position (machine position - measurement zero point (#2015 tlml-)) and program zero point offset (L).

Set the program zero point offset (L) by pressing the menu [Zero-pt offset] on the [Setup] - [T-meas] screen.





The tool length is automatically calculated by the following formula:

Tool length = Measurement position (machine coordinate value - measurement zero point (#2015 tlml-)) - Program zero point offset (L)

Note

(1) The measurement zero point is always the radius value regardless of the diameter/radius command.

(2) For the program zero point offset (L), set the diameter value for a diameter axis (#1019 dia = 1) or the radius value for a radius axis (#1019 dia = 0).

[Workpiece coordinate system offset designation method (#11087 = 2)]

In this method, the measurement zero point differs depending on the setting of the parameter "#1284 ext20/bit2".

<When "#1284 ext20/bit2" is set to "0">

The measurement zero point is the position shifted by the amount of the selected workpiece coordinate system offset from the machine coordinate zero point.

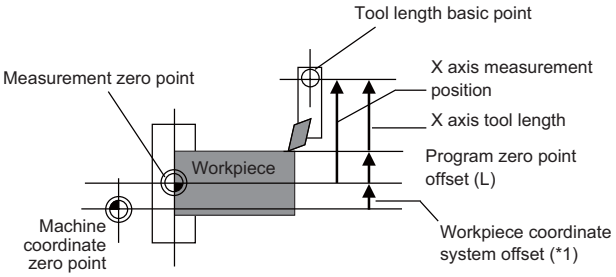
<When "#1284 ext20/bit2" is set to "1">

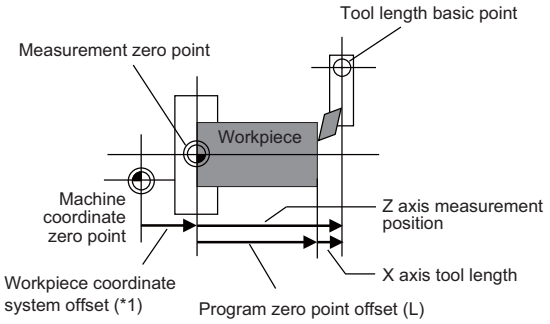
The measurement zero point is the position shifted by the total amount of the following offsets from the machine coordinate zero point.

- Currently selected workpiece coordinate system offset
- External workpiece coordinate offset
- Workpiece coordinate system shift
- Workpiece base point offset

Move the tool to the face or side, and calculate the tool compensation amount from the measurement position (workpiece coordinate position) and program zero point offset (L).

Set the program zero point offset (L) by pressing the menu [Zero-pt offset] on the [Setup] - [T-meas] screen.





(*1) When the parameter "#1284 ext20/bit2" is set to "1", it is the total amount of workpiece coordinate system offset, external workpiece coordinate offset, workpiece coordinate system shift and workpiece base point offset.

The tool length is automatically calculated by the following formula:

Tool length = Measurement position (workpiece coordinate position) - Program zero point offset (L)

Note

(1) For the program zero point offset (L), set the diameter value for a diameter axis (#1019 dia = 1) or the radius value for a radius axis (#1019 dia = 0).

(2) When the parameter "#1284 ext20/bit2" is set to "0", if the tool compensation amount is measured during automatic operation, the measurement zero point is the position shifted by the total amount of the following offsets from the machine coordinate zero point.

- Currently selected workpiece coordinate system offset
- External workpiece coordinate system offset
- G92 offset amount
- Local coordinate offset
- Workpiece coordinate system shift
- Workpiece base point offset

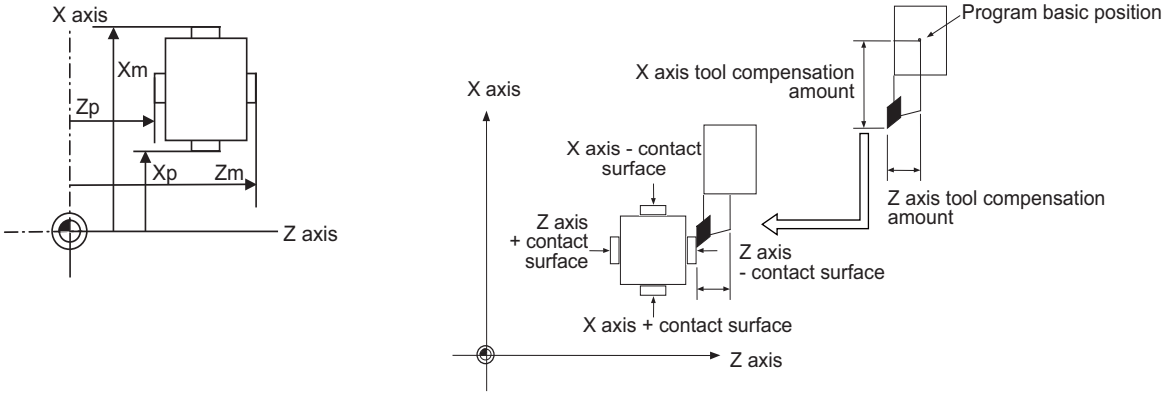
(3) When the parameter "#1284 ext20/bit2" is set to "0", the workpiece base point offset will not be added while the tool compensation amount is being measured even if a value is set in the parameter "#2701 Work base ofs" (Workpiece base point offset).

For L system, however, the workpiece base point offset is enabled during operation regardless of the setting of the parameter "#1284 ext20/bit2". This means that even if the workpiece base point offset is not added to the tool compensation offset amount, the workpiece coordinate zero point is shifted by the set amount of the workpiece base point offset.

■Manual tool length measurement 2

In an environment where the device equipped with a touch probe is used, the tool compensation amount can be calculated just by contacting the tool nose against the touch probe with manual feed, and it is stored to NC memory.
Set the operation mode to "JOG" or "HANDLE" to perform the measurement.
Set the measurement basic position (sensor position) with the parameters "#2015 tlml-" and "#2016 tlml+" when using the main spindle side, "#2097 tlml-" and "#2098 tlml+" when using the sub spindle side.

[Setting the measurement basic position]



When tool presetter operation is performed at the main spindle side:

Xm: Machine coordinate position of - sensor for X axis (position measured by moving in - direction) -> #2015 tlml- X axis

Zm: Machine coordinate position of - sensor for Z axis (position measured by moving in - direction) -> #2015 tlml- Z axis

Xp: Machine coordinate position of + sensor for X axis (position measured by moving in + direction) -> #2016 tlml+ X axis

Zp: Machine coordinate position of + sensor for Z axis (position measured by moving in + direction) -> #2016 tlml+ Z axis

When tool presetter operation is performed at the sub spindle side:

Xm: Machine coordinate position of sub-side - sensor for X axis (position measured by moving in - direction) -> #2097 tlml2- X axis

Zm: Machine coordinate position of sub-side - sensor for Z axis (position measured by moving in - direction) -> #2097 tlml2- Z axis

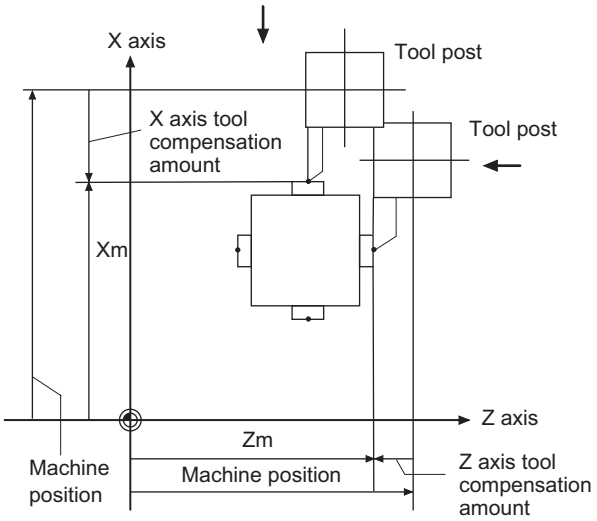
Xp: Machine coordinate position of sub-side + sensor for X axis (position measured by moving in + direction) -> #2098 tlml2+ X axis

Zp: Machine coordinate position of sub-side + sensor for Z axis (position measured by moving in + direction) -> #2098 tlml2+ Z axis

When an additional axis is measured, the axis to be set in the parameter "#2015 tlml-"/"#2016 tlml+" depends on the MTB specifications. (Parameters "#1281 ext17/bit1" (Selection of additional tool offset axis), "#1520 Tchg34" (Additional axis tool compensation operation))

The tool compensation amount is automatically calculated by the following formula.

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



When the tool touches the touch probe 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.

Note

(1) The compensation No. cannot be selected by using [↑], [↓] or page up/down keys while the measurement switch is being ON. Operations such as selecting an axis, and moving cursor to the wear data or the tool length data, are available by using said keys.

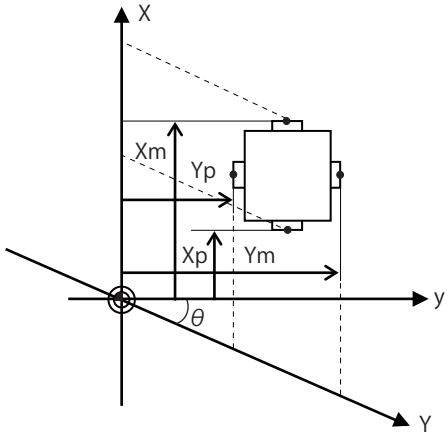
■Manual tool length measurement 2 (When this function is used with the inclined axis control)

During the inclined axis control function, perform the tool length measurement of basic axis and inclined axis as the following procedures.

[Setting the measurement reference position]

The measurement reference position (sensor position) is set on the sensor position within the orthogonal coordinate, not the position within the machine coordinate.

<Example of when X axis is the basic axis and Y axis is the inclined axis>



X: Actual axis
Y: Actual axis
y: Orthogonal coordinate
θ: Inclined angle

When tool presetter operation is performed at the main spindle side:

Xm: Orthogonal coordinate position of - sensor for X axis (position measured by moving in - direction) -> #2015 tlml- X axis

Ym: Orthogonal coordinate position of - sensor for Y axis (position measured by moving in - direction) -> #2015 tlml- Y axis

Xp: Orthogonal coordinate position of + sensor for X axis (position measured by moving in + direction) -> #2016 tlml+ X axis

Yp: Orthogonal coordinate position of + sensor for Y axis (position measured by moving in + direction) -> #2016 tlml+ Y axis

When tool presetter operation is performed at the sub spindle side:

Xm: Orthogonal coordinate position of sub-side - sensor for X axis (position measured by moving in - direction) -> #2097 tlml2- X axis

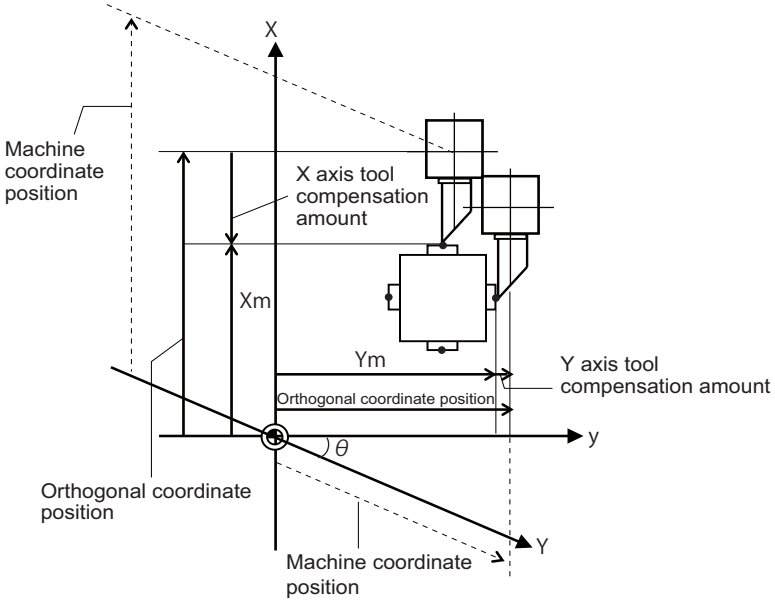
Ym: Orthogonal coordinate position of sub-side - sensor for Y axis (position measured by moving in - direction) -> #2097 tlml2- Y axis

Xp: Orthogonal coordinate position of sub-side + sensor for X axis (position measured by moving in + direction) -> #2098 tlml2+ X axis

Yp: Orthogonal coordinate position of sub-side + sensor for Y axis (position measured by moving in + direction) -> #2098 tlml2+ Y axis

The tool compensation amount is automatically calculated by the following formula.

Tool compensation amount = Orthogonal coordinate position - Measurement reference position (sensor position)



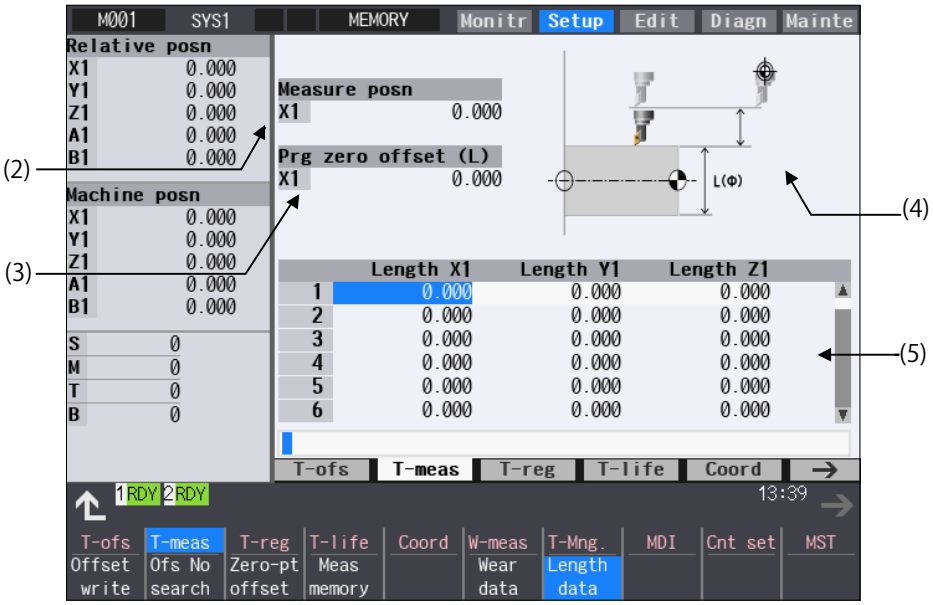
X: Actual axis
Y: Actual axis
X-y: Orthogonal coordinate
θ: Inclined angle

Two inclined axes move at the same time on orthogonal coordinates (X-y). The message "TLM axis is illegal" appears.

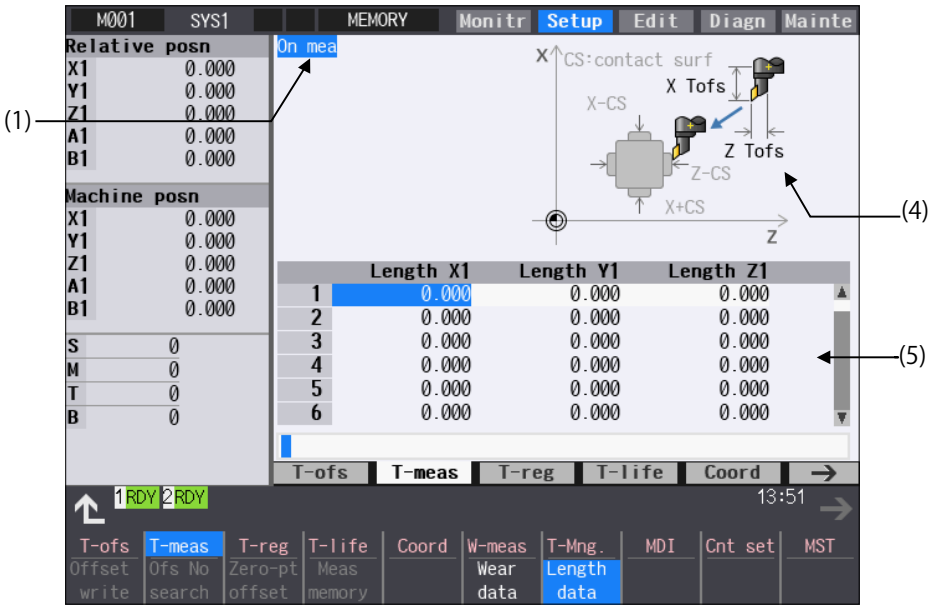
To measure the inclined axes, turn ON the Inclined axis control signal (YC36), and move them independently to touch the touch probe.

■Screen image

[Manual tool length measurement 1]

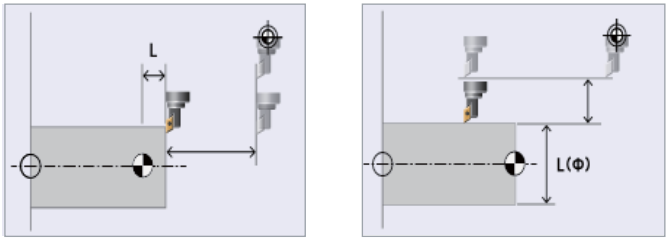
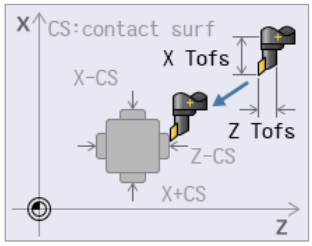


[Manual tool length measurement 2]



Display items

Display items	Details
(1) On mea	"On mea" is displayed when the measurement switch is ON. <Note> ♦This is displayed only for the manual tool length measurement 2.
(2) Measure posn	When the manual tool length measurement 1 is performed, the measurement result and measurement axis are displayed. <Normal operation mode> Measurement result is displayed by pressing the menu [Meas memory] when the cursor is placed at the axis to be measured. Measurement Position: Machine position when the menu operation is performed. <Simple operation mode> Measurement result for the axis selected using either the menu [Measure X] or the axis address key. Measurement Position: Machine position when the menu operation or address key operation is performed. <Note> ♦This is displayed only for the manual tool length measurement 1. ♦The data of maximum 3 axes displays when the manual tool length measurement 1 (Simple operation mode) is selected. ♦The axis names set in the parameter "#1022 axname2" are displayed.
(3) Prg zero offset (L)	Workpiece measurement result is set and displayed. Setting range: -99999.999 to 99999.999 (mm) <Normal operation mode> Setting is possible by pressing the menu [Zero-ptoffset] or touching the value area of the "Prg zero offset (L)". <Simple operation mode> Select the axis to measure using either the menu [Measure X] or the axis address key. Then the cursor moves to "Prg zero offset (L)", where setting a measured value is enabled. <Note> ♦The data of maximum 3 axes displays when the manual tool length measurement 1 (Simple operation mode) is selected.

Display items		Details
(4)	Guide graphic	Displays the measurement image. The contents of the guide drawing will differ depending on which measurement type (manual tool length measurement 1 or 2) is selected. <Manual tool length measurement 1> ♦Normal operation mode The guide drawing is switched to the axis pointed by the cursor in the tool compensation data. ♦Simple operation mode The guide drawing is switched to the selected axis, when the handle mode is selected. <Z axis> <Other axes>  <Manual tool length measurement 2> The guide drawing when using the touch probe is displayed. 
(5)	Tool compensation data	Displays the tool compensation data. <Note> ♦Cursor movement, data input and part system changing operation are same as "Tool compensation type III (L system)". ♦Automatically jumps to the compensation No. registered in "tool No. (tool length measurement 2)" (PLC device) during the manual tool length measurement 2. (Jumps even when the wear data is being displayed.) Note that the [↑], [↓], PAGE, and PAGE keys are disabled. ♦The screen displays the compensation data of four axes when the parameter "#12103 2nd add T-ofs ON" (2nd additional axis tool offset ON) is valid. ♦The tool compensation data of an axis which has been set to "1" (hide) by the parameter "#12117 T-ofs hide axis" is not displayed. ♦When the parameter "#1760 cfgPR10/bit6" (Wear compensation amount edit in data protection ON) is set to "1", wear compensation amount can be set regardless of whether the data protection key 1 is ON or OFF.

Note

(1) The values of "Measure posn" and "Prg zero offset (L)" will be cleared when changing of part systems, screens and measurement modes are carried out.

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

Menus (Manual tool length measurement 1 (normal operation mode) and manual tool length measurement 2)

Menus	Details
Offset write	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". Manual tool length measurement 1 Tool compensation amount = "Measure posn" value - "Prg zero offset (L)" value The values of "Meas posn" and "Prg zero offset (L)" will be cleared after compensation amount is written with the manual tool length measurement 1. <Note> ♦Tool compensation amount cannot be written in the following cases: - When the menu [Offset No search] and [Zero-pt offset] are highlighted - When the wear data is being displayed - When the measurement result exceeds the setting range ♦If the manual tool length measurement 2 is selected, this is grayed out and non-selectable. ♦When the parameter "#8924 Meas. confirm msg" is "1", this menu is highlighted and the operation 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 highlighting is canceled and the operation message is erased.
Ofs No search	When the compensation No. is set and the [INPUT] key is pressed, the tool compensation data with that number at its head appears. <Note> ♦If the measurement switch is turned ON, this is grayed out and non-selectable.
Zero-pt offset	This sets the "Prg zero offset (L)" value. If this menu is pressed, the cursor moves to the "Prg zero offset (L)" value. Point the cursor to the axis to be set, and set the data in the data setting area. Press the [INPUT] key to set the "Prg zero offset (L)" value. <Note> ♦If any mode other than the manual tool length measurement 1 is selected, this is grayed out and non-selectable.
Meas memory	Measurement result is calculated from the current machine position and displayed. <Note> ♦If any mode other than the manual tool length measurement 1 is selected, this is grayed out and non-selectable.
Wear data	Wear data is displayed in tool compensation data.
Length data	Tool length data is displayed in tool compensation data. This is the default mode when the power is initially turned ON.

Note

(1) The data displayed on the tool offset screen are not linked with the tool offset data on the tool measurement screen. (If the tool measurement screen is opened with tool wear data shown on the tool offset screen, the wear data will not always appear on the tool measurement screen.)

Menus (Manual tool length measurement 1 (simple operation mode))

Menus	Details
Offset write	The following value is written as the tool compensation amount, and the wear is cleared to "0". Tool compensation amount = "Measure posn" value - "Prg zero offset (L)" value •When the tool offset is written, the "Measure posn" and "Prg zero offset (L)" are cleared. The cursor moves to the tool compensation data. •The offset cannot be written in the following state. The operation message - "Can't write compensation data" is displayed. (a) When the menu [Ofs No search] is highlighted (b) During wear data display (c) When the measurement axis is not selected •If the tool compensation amount is outside the setting range, the offset writing fails. The operation message "Setting error" is displayed. <Note> •When the parameter "#8924 Meas. confirm msg" is set to "1", the menu "Offset write" is highlighted and confirmation message "OK?(Y/N)" is displayed. When you press the [Y] (or [INPUT]) key, the measurement position is written as the tool compensation amount. When you press any other key, the offset is not written. At this time, menu highlighting is canceled and the operation message is erased. If you press the menu [Offset write] again during operation message display, the reverse display is released and the operation message is also erased.
Ofs No search	When the compensation No. is set and the [INPUT] key is pressed, the tool compensation data with that number at its head appears. This operation will not change the column where the cursor is placed. If this operation is carried out with the cursor placed at "Prg zero offset (L)", the cursor will remain displayed at "Prg zero offset (L)".
Measure X	Select the axis to measure. The menu is highlighted when it is selected. Multiple axes can be selected. To cancel, press the menu key again or press [Cancel value]. (*1)
Measure Z	Select the axis to measure. The menu is highlighted when it is selected. Multiple axes can be selected. To cancel, press the menu key again or press [Cancel value]. (*1)
Measure C	Select the axis to measure. The menu is highlighted when it is selected. Multiple axes can be selected. To cancel, press the menu key again or press [Cancel value]. (*1)
Wear data	Wear data is displayed in tool compensation data.
Length data	Tool length data is displayed in tool compensation data. This mode is enabled when turning ON the power.
Cancel value	This cancels displaying the value of the measurement result and "Prg zero offset (L)". The cursor moves from "Prg zero offset (L)" to the tool compensation data. If any mode other than the manual tool length measurement 1 is selected, this menu will be grayed out and non-selectable.

(*1) The axis name to be displayed follows the parameter "#1022 axname2".

Note

(1) The data displayed on the tool offset screen are not linked with the tool offset data on the tool measurement screen. (If the tool measurement screen is opened with tool wear data shown on the tool offset screen, the wear data will not always appear on the tool measurement screen.)

Carrying out tool measurement (Manual tool length measurement 1)

■Normal operation mode

(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 "9.6 Manual Numerical Value Command" for details.)

(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.
- When the axis to apply the mirror image is to be measured, turn ON the mirror image.

(3) Move the tool to the side surface of the workpiece. (In the case of Z-axis move the tool to the end surface of the workpiece.)

(4) The menu [Meas memory] is pressed without retracting the tool. → Save the measurement result. Saved value is displayed in the measurement area.

<Note>

- For the axis to which the mirror image is turned ON, the measurement position considering the mirror status is displayed.

(5) Retract the tool and measure the workpiece. And then press the menu [Zero-ptoffset]. → The menu [Zero-ptoffset] is highlighted and the cursor moves to the program zero-point offset (L) value.

(6) Measure the workpiece and set the offset value from the tool position (side surface of the workpiece) to the program zero-point. → 10 is set for the "Prg zero offset (L)" value. Highlight of the menu [Zero-ptoffset] is canceled and the cursor moves to the tool compensation data.

10 [INPUT]

<Note>

- Compensation amount is written in diameter value for diameter command, or in radius value for radius command.

(7) Press the menu [Offset write]. → "Measurement result - workpiece measurement value" is set to compensation No.3, X axis.

<Note>

- The values of "Measure posn" and "Prg zero offset (L)" are cleared.

■Simple operation mode

(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 "9.6 Manual Numerical Value Command" for details.)

(2) Move the cursor to the position to input the measurement result (compensation No.3). → The cursor is displayed at compensation No.3.

<Note>

- When the axis to apply the mirror image is to be measured, turn ON the mirror image.

(3) Move the tool to the side surface of the workpiece. (In the case of Z-axis move the tool to the end surface of the workpiece.)

(4) Press the address key of the axis to measure or the menu [Measure X], without retracting the tool. → Save the measurement result. The memorized value is displayed at "Prg zero offset (L)", and the cursor moves to "Work meas".
The operation message "Input program Z-point offset" displays.

<Note>

- If multiple axes which have the same first letter of the axis name set by the parameter "#1022 axname2" exist, the axis address key treats the first found axis (in the order of 1st axis --> 2nd axis --> additional axis) as the measurement axis.
- For the axis to which the mirror image is turned ON, the measurement position considering the mirror status is displayed.

(5) To measure the other axes at the same time, repeat (3) to (4). → Save the measurement value of the selected axis. Saved value is displayed in the measurement area. The cursor will not move from the top line.

(6) Retract the tool and measure the workpiece.

(7) Measure the workpiece and set the offset value from the tool position (side surface of the workpiece) to the program zero-point.
(Example) 10 [INPUT] → The value 10 is set in the top line of "Prg zero offset (L)". If there are two or more axes to measure, the cursor moves down to the next line.

<Note>

- Compensation amount is written in diameter value for diameter command, or in radius value for radius command.

(8) To set the "Prg zero offset (L)" of the other axes, repeat (7).
(Example) 5 [INPUT] → The value "5" is set to the 2nd line of "Work meas".

<Note>

- Compensation amount is written in diameter value for diameter command, or in radius value for radius command.

(9) Press the menu [Offset write]. → The result of "Meas val - Prg zero offset (L)" will be set as the compensation amount for X and Z axes of the compensation No.3 (where the cursor is currently positioned).

<Note>

- The values of "Measure posn" and "Prg zero offset (L)" are cleared, and the highlighted menus ([Measure X] - [Measure C]) will return to original status.

Precautions for manual tool length measurement 1

(1) When compensation amount is written for an axis that has not completed zero point return, the operation message "TLM axis not returned to ref. position" will appear.

(2) When the parameter "#8924 Meas. confirm msg" is set to "1", the menu "Offset write" is highlighted and confirmation message "OK?(Y/N)" is displayed. When you press the [Y] (or [INPUT]) key, the measurement position is written as the tool compensation amount. When you press any other key, the offset is not written. At this time, menu highlighting is canceled and the operation message is erased.

(3) When the parameter "#1124 ofsfix" is set to "0", the cursor moves to the next offset No. However, when "#1124 ofsfix" is "1", the cursor does not move to the next offset No.

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

Carrying out tool measurement (Manual tool length measurement 2)

(1) Select the tool to be measured with manual numerical value command. (Refer to "9.6 Manual Numerical Value Command" for details.)

(2) Set the compensation No. of the compensation data to be set in "Tool No. (Tool length measurement 2)" (PLC device).

(3) Set the compensation No. of the wear data to be cleared in "wear compensation No. (tool presetter)" (PLC device).

(4) Turn ON the measurement switch.
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)

→ "On mea" is displayed.
Guide drawing for the manual tool length measurement 2 is displayed.
Tool compensation data is displayed, placing the compensation No. set in step (2) at the top.
<Note>
♦The row at the cursor position will not be changed.

(5) Move the measurement axis by manual feed and have the tool nose contact with the touch probe.

→ Measurement value is set for the measurement axis of the compensation No. set in step (2).

(6) Retract the tool.
To measure another axis, return to step (5).

(7) Turn the measurement switch OFF.

→ "On mea" disappears.

Note

(1) When the tool is touched to the sensor, one axis of the following can be moved; +X (1st axis), -X (1st axis), +Z (2nd axis), -Z (2nd axis), (+Y (additional axis), and -Y (additional axis)). If two axes are moved simultaneously, the measurement cannot be performed. The axes do not move because the contact surface of the touch probe cannot be distinguished. The message "TLM axis is illegal" appears.

(2) After contacting the sensor, 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 500 ms or more.

♦The tool nose is moved 100 μm or more after the sensor signal has turned OFF.

The above conditions are set with parameter "#1227 aux11/bit2" (measures against tool setter chattering).

6.4 Setting Tool Compensation Amount (Tool Offset Screen)

Setup – Offset (M system)　　Monitr – T-ofs (M system)

Setup – Offset (L system)　　Monitr – T-ofs (L system)

The tool compensation data can be set and displayed tool offset (T-ofs) screen of the setup (Setup) screen.

For M system, three types of tool compensation can be switched by the setting of the parameters "#1037 cmdtyp" and "#1046 T-ofs disp type".
- Tool compensation type I (#1037=1/#1046=0)
- Tool compensation type II (#1037=2/#1046=0)
- Tool compensation type III (#1046=1)

For L system, tool compensation type III is selected regardless of the setting of the parameter #1037/#1046".
The tool radius compensation can be carried out using the diameter value if the parameter "#8117 OFS Diam DESIGN" is set to "1".
The number of tool compensation sets to be designated or displayed varies depending on the optional specifications.

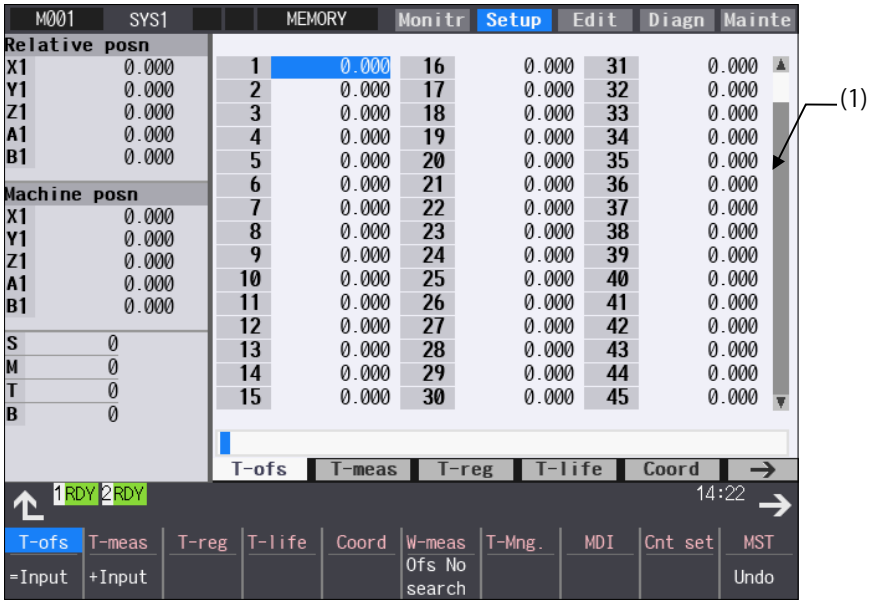
Tool compensation data can be set or displayed on the monitor (Monitr) screen.

Note

(1) 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.

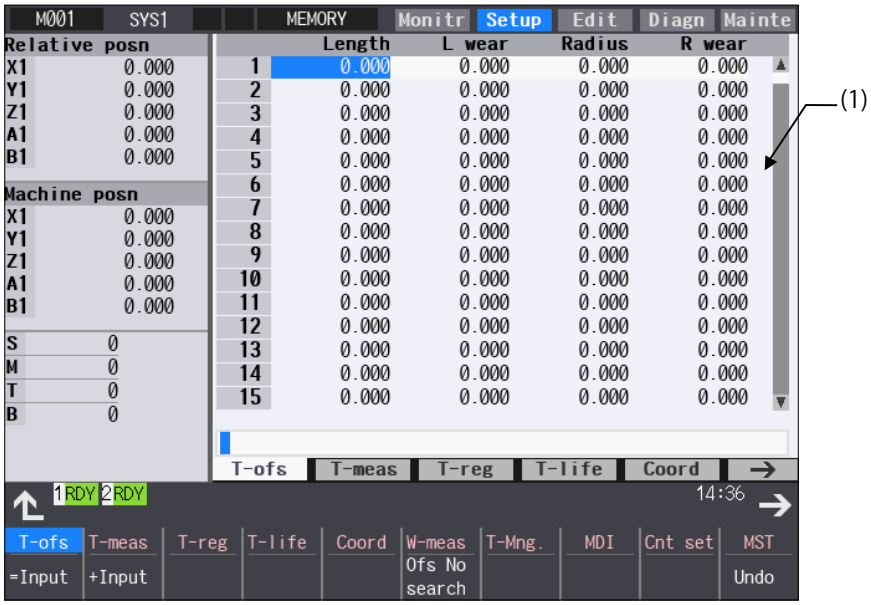
[Tool compensation type I] (M System): Parameter "#1037 cmdtyp" = 1

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 3-dimensional tool radius compensation.



[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 3-dimensional tool radius compensation (compensation vector designation).



[Tool compensation type III] (M/L System): Parameter "#1037 cmdtyp" = 3

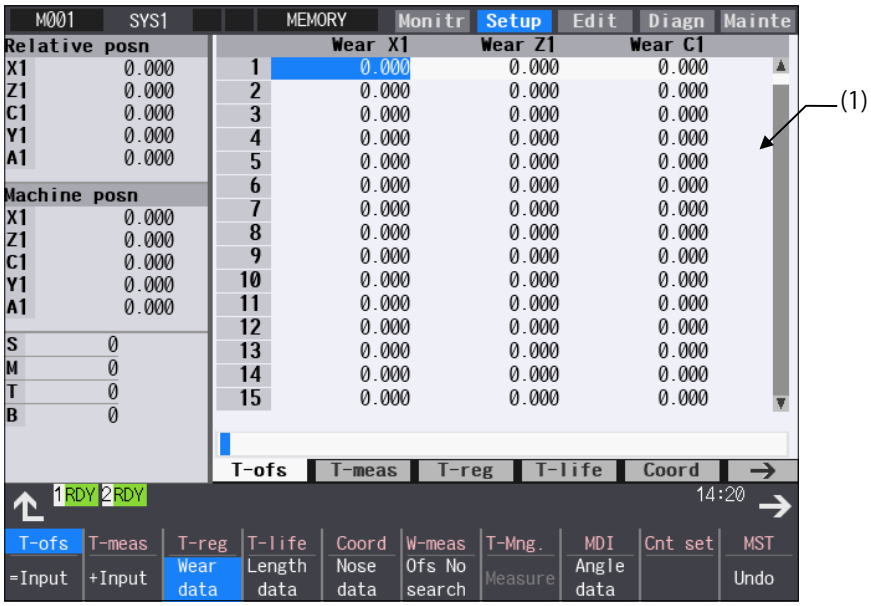
This type has 4 kinds of display such as wear data, tool length data, nose data and cutting-edge angle data.

The wear data and the tool length data for four axes can be set and displayed by the parameter setting of "#12103 2nd add T-ofs ON". Specified axis is set not to be set and displayed by the parameter setting of "#12117 T-ofs hide axis."

■Wear data

Set the wear amount of nose for each tool you use.

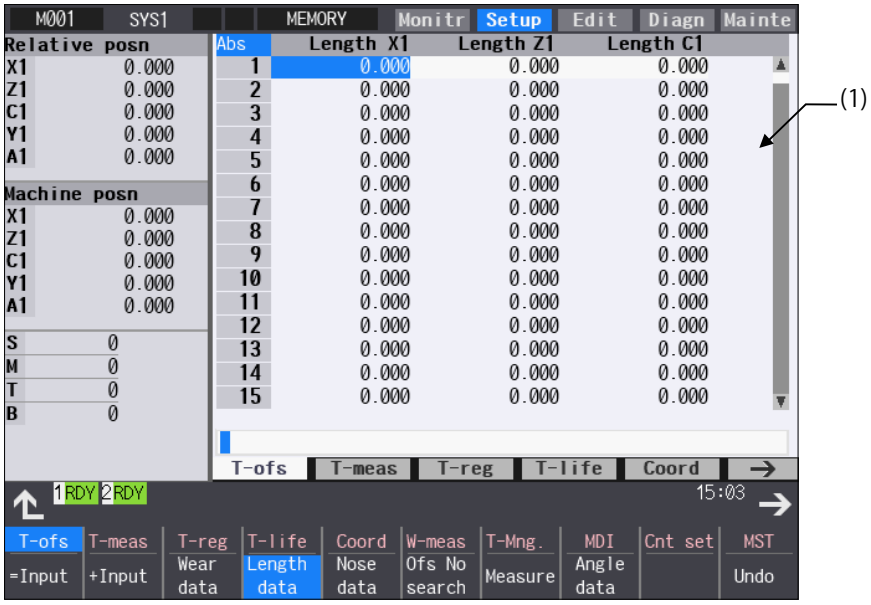
When a tool compensation number is designated by the tool command (T command), the compensation operation is performed with the tool length data, nose data and cutting-edge angle data.



■Tool length data

Set the tool length according to the reference position of program for each tool used.

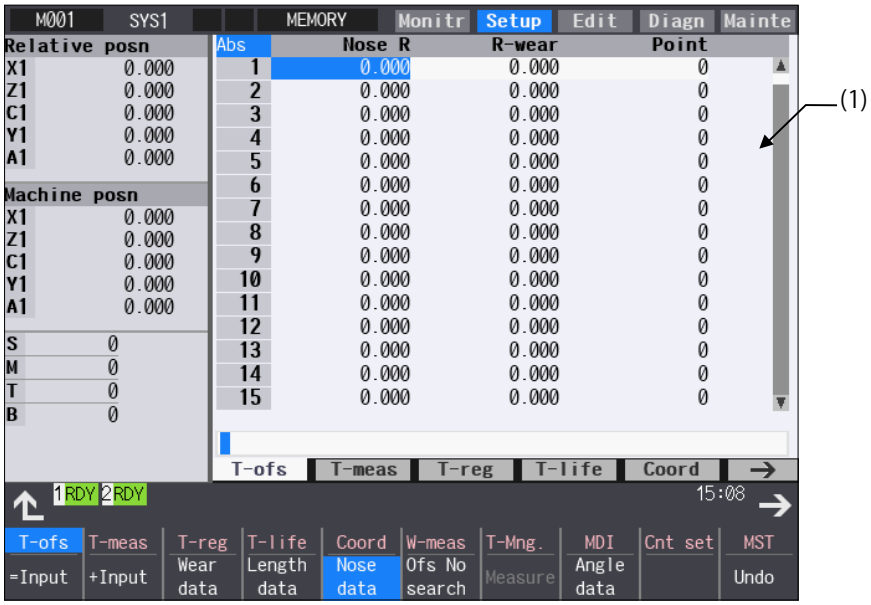
When a tool compensation number is designated by the tool command (T command), the wear data is offset together with the wear data, nose data and cutting-edge angle data.



■Nose data

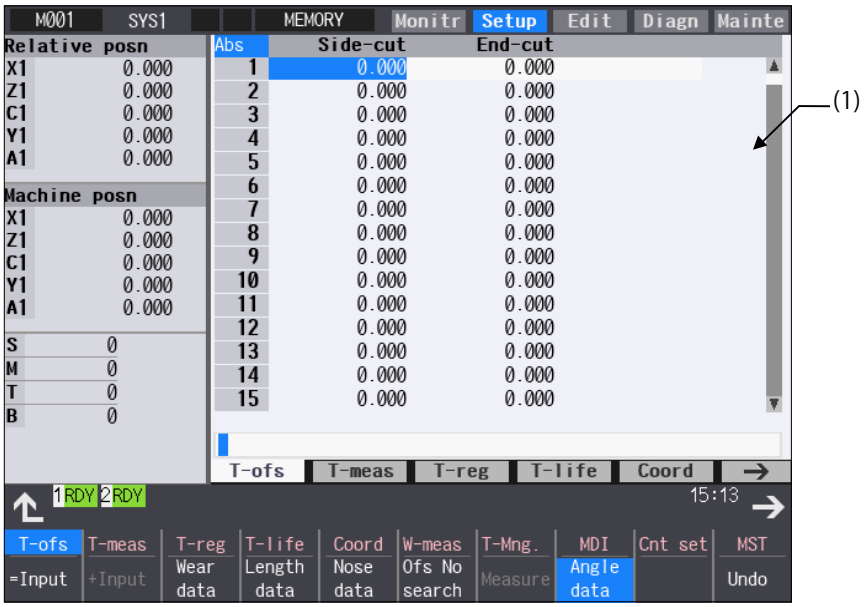
Set the radius (Nose R), wear radius (R-wear), and point (Point) of the nose that is mounted on the tool for each tool used.

When a tool compensation number is designated by the tool command (T command), the compensation operation is performed with the tool length data, wear data and cutting-edge angle data.



■Cutting-edge angle data

Set the side-cutting-edge angle and end-cutting-edge angle for each tool to be used. When a tool compensation number is designated by the tool command (T command), the compensation operation is performed with the tool length data, wear data and cutting-edge angle data during compound type fixed cycle for turning machining I.



> **⚠ 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 items	Details
(1) Display area	Displays the tool compensation data. The cursor is displayed, and you can set the data. The data exceeding the display area can be displayed by the following keys. [↑] [↓] : Scroll one line at a time. [PAGE] [PAGE] : Switches the display contents for each number of compensation data sets. Compensation number: This is a compensation data number. The other display items change depending on the tool compensation type. Refer to the following table for details.

<Display items and setting range>

The setting range here is the value when the minimum command unit is 1μm (#1003 = B, #1041 = 0).

Type	Display item	Details	Setting range
I	Compensation data	Sets the tool length compensation amount. (*5)	-9999.999 to 9999.999
II	Length	Sets the tool length compensation amount. (*5)	
	L wear	Sets the compensation amount of tool length wear. (*5)	
	Radius	Sets the tool radius compensation amount. (*5)	
	R wear	Sets the tool radius wear compensation amount. (*5)	
III	Wear data	<L system> Sets the wear compensation amount of 1st axis, 2nd axis, additional axis, and 2nd additional axis. (*1)(*2)(*5)(*7)(*8)(*10) <M system> Sets the wear compensation of the axes designated by the parameters "#1026 base_I", "#1027 base_J" and "#1028 base_K". (*5)(*9)(*10)	
	Tool length data	<L system> Sets the tool length compensation amount of 1st axis, 2nd axis, additional axis, and 2nd additional axis. (*1)(*2)(*7)(*8)(*10)(*11) <M system> Sets the tool length compensation amount of the axes designated by the parameters "#1026 base_I", "#1027 base_J" and "#1028 base_K". (*9)(*10)(*11)	
	Nose R (*3)(*11)	Sets the tool radius (Nose R).	
	R wear (*3)(*6)	Sets the tool radius (Nose R) wear.	
	Point (*4)	Sets the nose point (number).	0 to 9
	Side-cut (*4)	Sets the side-cutting-edge angle.	0.000 to 180.000
	End-cut (*4)	Sets the end-cutting-edge angle.	0.000 to 180.000

(*1) Select the additional axis by specifying the parameter (#1281 ext17 bit1/#1520 Tchg34).

(*2) If the number of axes in the part system is 2 or less, the name of non-existing axis is displayed according to the parameter setting as follows. Setting the compensation amount to non-existing axis does not mean anything.

If "#1281 (PR) ext17/bit1" (Tool offset additional axis selection) is set to "0":
The axis name is not displayed.

If "#1281 (PR) ext17/bit1" (Tool offset additional axis selection) is set to "1":
If the 2nd axis does not exist, the axis name designated by "#1028 base_K" is displayed. If the additional axis does not exist, the axis name designated by "#1027 base_J" is displayed.

(*3) If parameter "#1019 dia" (Diameter command) is set to "0", set the radius. If it is set to "1", set the diameter.

(*4) The nose point, side-cutting-edge angle and end-cutting-edge angle are entered as the absolute position even if you press the menu [+ Input].

(*5) It may differ from the setting range in the table depending on the setting value of "#8010 ABS. MAX.", "#8011 INC. MAZ." or range setting screen.
When "n" is set to #8010, the range of "-n" to "n" can be set as compensation amount. When a value exceeding the setting value of #8010 or when a compensation data which is out of the range set on the range setting screen is input, the message "Setting error" appears.

(*6) It may differ from the setting range in the table depending on the setting value of "#8010 ABS. MAX." or range setting screen.
When "n" is set to #8010, the range of "-n" to "n" can be set as compensation amount. When a value exceeding the setting value of #8010 or when a compensation data which is out of the range set on the range setting screen is input, the message "Setting error" appears.

(*7) The axis names set by the parameter "#1022 axname2" are displayed for 1st axis, 2nd axis, additional axis, and 2nd additional axis.

(*8) The 2nd additional axis can be set or displayed when the parameter "#12103 2nd add T-ofs ON" is set to "1". The axis name is not displayed when a non-existing axis is designated by the parameter "#12104 2nd add T-ofs".

(*9) The 2nd axis names set by "#1022 axname2" are displayed for the axes set in the parameters "#1026 base_I", "#1027 base_J" and "#1028 base_K". However, if any axes designated by the parameter "#1026 base_I" to "#1028 base_K" does not exist, the axis name of the parameter "#1026 base_I" to "#1028 base_K" will be displayed instead.

(*10) The tool compensation data of an axis which has been set to "1" (hide) by the parameter "#12117 T-ofs hide axis" is not displayed.

(*11) It may differ from the setting range in the table depending on the setting value of the range setting screen.
When a compensation data which is out of the range set on the range setting screen is input, the message "Setting error" appears.

The setting ranges are as follows based on the combination of "#1003 iunit" (Input unit) and "#1041 Initial inch".

#1003 iunit	#1041 Initial inch	Setting range	
		Type II (Length wear/Radius wear) Type III (Wear data/Radius wear)	Type I Type II (Length dimension/Radius dimension) Type III (Tool length data/Nose R)
B	0	-9999.999 to 9999.999	-9999.999 to 9999.999
	1	-999.9999 to 999.9999	-999.9999 to 999.9999
C	0	-9999.9999 to 9999.9999	-9999.9999 to 9999.9999
	1	-999.99999 to 999.99999	-999.99999 to 999.99999
D	0	-9999.99999 to 9999.99999	-9999.99999 to 9999.99999
	1	-999.999999 to 999.999999	-999.999999 to 999.999999
E	0	-9999.999999 to 9999.999999	-9999.999999 to 9999.999999
	1	-999.9999999 to 999.9999999	-999.9999999 to 999.9999999

When data without the decimal point is input, a set unit can be specified by the parameter "#8119 Comp. unit switch".

The standard settings and display ranges are as follows.

Type	Display position
I	Top of display area (*1)
II	Left side of radius (*1) Left side of radius wear (*1)
III	<L system> Left side of axis name (*2) <M system> Nose R, left side of radius wear (*1)

(*1) It is displayed when the parameter "#8117 OFS Diam DESIGN" is "1".

(*2) It is displayed at the left of axis name when the parameter "#1019 dia" (Diameter command) is set to "1".

Menus

Menus	Details
=Input	This executes an absolute input. When "#8929 Disable=INPUT:comp" is set to "1", this menu cannot be selected.
+Input	This executes an incremental input. <Note> •This menu is disabled while the cutting-edge angle data is displayed.
Wear data	This switches to the wear data display. <Note> •This menu is displayed for type III only.
Length data	This switches to the tool length data display. <Note> •This menu is displayed for type III only.
Nose data	This switches to the nose data display. <Note> •This menu is displayed for type III only.
Ofs No search	When the compensation No. is set and the [INPUT] key is pressed, the tool compensation data with that number at its head appears. The cursor moves to the contents of that first line.
Measure	Measure the tool compensation amount. <Note> •This menu is displayed only for type III, and is valid for "L system" and "tool length data display". •When the parameter "#1235 set07/bit5" (Tool wear data clear on tool length measurement) is set to "1", the wear amount of measured axis is cleared to "0".
Angle data	This switches to the cutting-edge angle data display. <Note> •This menu is displayed for type III only. This menu is disabled for M system.
Undo	This undoes the last changes to the tool compensation data. (This menu is valid for the "Data Input", "Paste Line" and "Undo" operations.) <Note> •When an operation parameter "#8939 Undo confirm msg" is set to "1", the menu is highlighted while the operation message is displayed.
Line copy	This copies the contents of the tool compensation data in the selected line (one line).
Line paste	The contents (one line) of the copied tool compensation data are written to the line where the cursor is. If the copied data is changed after the menu [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.
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. (Example) 1/E: Set all compensation data to "0". If the [INPUT] key is pressed without selecting a line, the compensation data in the line where the cursor is currently positioned is erased.
Abs/Inc	Switch the setting method (absolute/incremental input) for the tool compensation data by [INPUT] key. <Note> •When a parameter "#8941 ABS/INC for T-ofs" is set to "0", it is fixed to an absolute input.

With type III, wear data is displayed when the power is turned ON.

After turning the power ON, the display selection for wear data, length data, nose data and cutting-edge angle data are retained.

6.4.1 Setting the Tool Compensation Data

Operation method (Setting "10.000" in the length wear data of compensation No. (102))

(1) Press the menu [Ofs No search].

(2) Designate the compensation No.
102 [INPUT]
The cursor can be moved to 102 using the [↑], [↓], PAGE, or PAGE keys.

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

(4) Input the length wear value.
10 [INPUT]
The value can also be designated by pressing the menu [=Input] instead of the [INPUT] key.

Note

(1) If the parameter "#11017 T-ofs set at run" is set to "1", the tool compensation amount data during automatic operation or during pause of operation can be set.

(2) When the parameter "#8929 Disable=INPUT:comp" is set to "1", the menu [=Input] will be grayed.

(3) This setting may be disabled due to the parameter "#8933 Disable lngth comp" or "8934 Disable wear comp" setting. For details, refer to "6.4.6 Disabling the Setting of Tool Compensation Amount".

(4) The above shows the setting of the tool compensation data when the parameter "#8941 ABS/INC for T-ofs" is set to "0". For details, refer to "6.4.2 Switching the Input Method for the Tool Compensation Data".

(5) Any of the menu [=Input], [+Input] or [INPUT] key is pressed when the parameter "#1124 ofsfix" is set to "0", the cursor will move to the next compensation No.

(6) When the parameter "#1760 cfgPR10/bit6" (Wear compensation amount edit in data protection ON) is set to "1", wear compensation amount can be set regardless of whether the data protection key is ON or OFF.

Operation method ("0.012" is calculated and set in the length wear data of compensation No. 102)

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

(2) Input a numeric value.
0.012 [+Input]



The input value added to the original value is displayed.
(Example) When the original value is 10.000, the calculation formula is as follows:
10.000 + 0.012 = 10.012

Note

(1) If the parameter "#11017 T-ofs set at run" is set to "1", the tool compensation amount data during automatic operation or during pause of operation can be set.

(2) This setting may be disabled due to the parameter "#8933 Disable lngth comp" or "8934 Disable wear comp" setting. For details, refer to "6.4.6 Disabling the Setting of Tool Compensation Amount".

(3) Any of the menu [=Input], [+Input] or [INPUT] key is pressed when the parameter "#1124 ofsfix" is set to "0", the cursor will move to the next compensation No.

(4) When the parameter "#1760 cfgPR10/bit6" (Wear compensation amount edit in data protection ON) is set to "1", wear compensation amount can be set regardless of whether the data protection key is ON or OFF.

(5) When entering data without decimal points, you can select the 1-mm (or 1-inch) data input or the input in the minimum input unit (same as "#1003 iunit") by designating the "#8119 Comp. unit switch" parameter.

<1-mm unit: "#8119 Comp. unit switch" is "0".>

Input		Result
1		1.000
100		100.000
10.		10.000
.012		0.012

<Minimum input unit (#1003 iunit B): "#8119 Comp. unit switch" is "1".>

Input		Result
1		0.001
100		0.100
10.		10.000
.012		0.012

Operation method (Input multiple tool compensation data at a time)

(Example) Setting "10.000" in "X1" and "20.000" in "Z1" for respective tool length data of the compensation No. 102

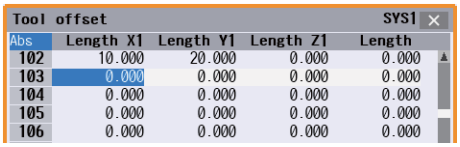
(1) Press the menu [Ofs No search].

(2) Designate the compensation No.
102 [INPUT]
The cursor can be moved to 102 using the [↑], [↓], PAGE or PAGE key.

→ The designated No. is displayed at the top of the compensation No. area.

(3) Set a value.
Input the data for each column dividing with "/".
1st column/2nd column/3rd column/4th column
10/20 [INPUT]
The value can also be set by pressing the menu [=Input] instead of the [INPUT] key.

→ The designated values are set.



Note

(1) If the parameter "#11017 T-ofs set at run" is set to "1", the tool compensation amount data during automatic operation or during pause of operation can be set.

(2) When the parameter "#8929 Disable=INPUT:comp" is set to "1", the menu [=Input] will be grayed.

(3) This setting may be disabled due to the parameter "#8933 Disable lngth comp" or "8934 Disable wear comp" setting. For details, refer to "6.4.6 Disabling the Setting of Tool Compensation Amount".

(4) The above shows the setting of the tool compensation data when the parameter "#8941 ABS/INC for T-ofs" is set to "0". For details, refer to "6.4.2 Switching the Input Method for the Tool Compensation Data".

(5) Any of the menu [=Input], [+Input] or [INPUT] key is pressed when the parameter "#1124 ofsfix" is set to "0", the cursor will move to the next compensation No.

(6) Compensation data for the number of the columns currently being displayed can be set at a time.

(7) When you input multiple tool compensation data at a time, the first data will be set at the leftmost column of the selected line regardless of the current cursor position.

(8) Inputting of multiple tool compensation data is available for the tool compensation type II and III.

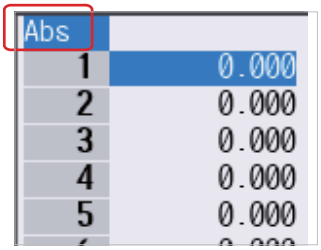
(9) When you input multiple tool compensation data at a time with the tool compensation type III, the data of the axis which is set not to be displayed (parameter "#12117 T-ofs hide axis" is set to "1") is skipped.
For example, when the 2nd column is set not to be displayed, input the data with the following format to set the tool compensation data for the 1st column and the 3rd column at a time.

1st column/3rd column/4th column
30/40 [INPUT]

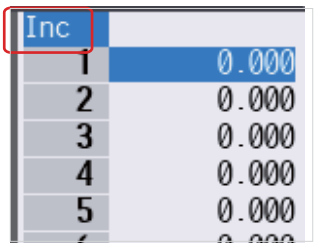
6.4.2 Switching the Input Method for the Tool Compensation Data

When a parameter "#8941 ABS/INC for T-ofs" is set to "1", the absolute/incremental input by the [INPUT] key can be switched. When attempting to switch it, the current input mode is displayed on the upper left of the tool compensation amount. The absolute/incremental input can be switched either by the short cut key ([I] key) or by pressing the menu [Abs/Inc]. The chosen method is kept after switching the screen and turning the power OFF.

■Absolute input mode



■Incremental input mode



6.4.3 Erasing the Tool Compensation Data

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

(1) Move the cursor to the data to be erased using the [↑], [↓], PAGE, or PAGE key.

(2) Press the menu [Line clear] and the [INPUT] key.

(3) Press the [Y] or [INPUT] key.
To cancel the erasing, press a key other than [Y] or [INPUT].
➡ The selected line of data (one line amount) is cleared to "0".
The cleared data is displayed at the top of the compensation data.

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

(1) Press the menu [Line clear].

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

(3) Press the [Y] or [INPUT] key.
To cancel the erasing, press a key other than [Y] or [INPUT].
➡ The compensation data for the selected compensation number is erased, and the menu highlight returns to normal.
The cleared data is displayed at the top of the compensation data.

Note

(1) If the [INPUT] key is pressed without selecting an compensation number, the line where the cursor is located is erased.
(2) If the parameter "#11017 T-ofs set at run" is set to "1", the tool compensation amount data during automatic operation or during pause of operation can be erased.
(3) This setting may be disabled due to the parameter "#8933 Disable lngth comp" or "8934 Disable wear comp" setting. For details, refer to "6.4.6 Disabling the Setting of Tool Compensation Amount".

6.4.4 Copying and Pasting the Tool Compensation Data

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

(1) Move the cursor to the data to be copied using the [↑], [↓], PAGE or PAGE key.

(2) Press the menu [Line copy].

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

(4) Press the menu [Line paste]. → 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.

Note

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

(2) In tool compensation type lll, the copied data cannot be pasted to the different item (Wear/Length/Nose).

(3) If the parameter "#11017 T-ofs set at run" is set to "1", the tool compensation amount data during automatic operation or during pause of operation can be erased.

(4) This setting may be disabled due to the parameter "#8933 Disable lngth comp" or "8934 Disable wear comp" setting. For details, refer to "6.4.6 Disabling the Setting of Tool Compensation Amount".

6.4.5 Undoing Change of Tool Compensation Data

(1) Move the cursor to the data to be set using the [↑], [↓], PAGE, or PAGE key.

(2) Input a numeric value.
(Example) 10 [INPUT]

(3) Press the menu [Undo]. → The value is returned to the previous value.

Note

(1) This setting will be disabled due to the parameter "#8933 Disable lngth comp" or "#8934 Disable wear comp" setting. For details, refer to "6.4.6 Disabling the Setting of Tool Compensation Amount".

(2) When "#8939 Undo confirm msg" is set to "1", the menu [Undo] is highlighted and confirmation message "OK?(Y/N)" is displayed. The program returns to the previous status by pressing [Y] or [INPUT] key. If any key other than [Y] or [INPUT] is pressed, the operation to undo will be canceled. When the operation is canceled, the menu highlight returns to normal and the message disappears.

(3) When the program does not return to the previous status by pressing [Undo], [Undo] is not highlighted and the operation message "Data protect" or "Cannot return to origin" appears.

(4) The menu [Undo] cannot be used to clear multiple lines.

(5) When tool length is measured with the parameter "#1235 set07/bit5" set to "1", the length cannot be reset to the previous value by using the menu [Undo].

6.4.6 Disabling the Setting of Tool Compensation Amount

Depending on the setting of the parameter "#8933 Disable lngth comp"/"#8934 Disable wear comp", the setting of the shape compensation amount/wear compensation amount is disabled.
Depending on the tool compensation type, different operation will be disabled.

Tool compensation type		"#8933 Disable lngth comp" = "1"	"#8934 Disable wear comp" = "1"
Type I	----	The menus [=Input], [+Input], [Line copy], [Line paste], [Line clear], and [Undo] are grayed out and cannot be selected. Any setting by [INPUT] key is disabled.	----
Type II	----	The menus [Line copy], [Line paste], and [Line clear] are grayed out and cannot be selected. The length and radius setting by [=Input], [+Input], and [INPUT] keys are disabled.	The menus [Line copy], [Line paste], and [Line clear] are grayed out and cannot be selected. The length wear and radius wear setting by [=Input], [+Input], and [INPUT] keys are disabled.
Type III	"Wear"	----	The menus [=Input], [+Input], [Line copy], [Line paste], [Line clear], and [Undo] are grayed out and cannot be selected. Any setting by [INPUT] key is disabled.
	"Length"	The menus [=Input], [+Input], [Measure], [Line copy], [Line paste], [Line clear], and [Undo] are grayed out and cannot be selected. Any setting by [INPUT] key is disabled.	----
	"Nose"	The menus [Line copy], [Line paste], and [Line clear] are grayed out and cannot be selected. The nose R setting by [=Input], [+Input], and [INPUT] keys are disabled.	The menus [Line copy], [Line paste], and [Line clear] are grayed out and cannot be selected. The R wear setting by [=Input], [+Input], and [INPUT] keys are disabled.
	"Cutting-edge angle"	The menus [=Input], [Line copy], [Line paste], [Line clear] and [Undo] are grayed out and cannot be selected. Any setting by [INPUT] key is disabled.	----

6.4.7 Measuring Tool Compensation Amount

The measurement method will differ depending on the setting of the parameter "#11087 Meas basic point".

[Note on the wear amount]

When the parameter "#1235 set07/bit5" (Tool wear data clear on tool length measurement) is set to "1", the wear amount of measured axis is cleared to "0".

Measuring the tool compensation amount of Z axis and X axis (diameter axis) with the machine zero point designation method (#11087 = 0)

(1) Select the tool to be measured with manual numerical value command. (Refer to “9.6 Manual Numerical Value Command" for details.)

(Example) T1003

(2) Displays the tool compensation amount screen.

Menu [Offset]

→ The tool compensation amount screen is displayed. The tool compensation number of the selected tool is displayed at the top, and the cursor is located on the first line.

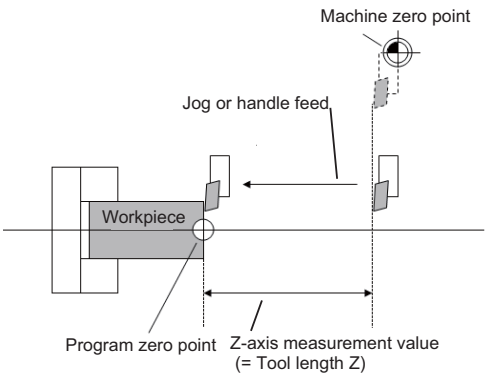
<Note>

- If the parameter "#8972 T code offset disp" is set to "0", the tool compensation number of selected tool is not displayed at the top. Locate the cursor onto the tool compensation number of the tool to be measured.
- When the axis to apply the mirror image is to be measured, turn ON the mirror image.

(3) Press the menu [Length data].

(4) Select the "Manual handle feed" mode or "Jog feed" mode.

(5) Select the Z axis, and manually move the tool to the work end face (program origin).



(6) Enter the name of axis to be measured, and the measurement value (radius) of workpiece.

Z0.

When you enter "Z", the cursor is moved to the Z axis column.

<Note>

- If you input an axis name that does not exist, the cursor does not move.
- The second and subsequent characters are not considered to be the axis name and, therefore, the cursor does not move. (Example: If you enter "XZ", the cursor moves on the X axis, but the cursor does not move on the Z axis.)
  Also, if anything other than tool length data is displayed, the cursor does not move even when the first input value is the axis name.
- If multiple axes which have the same first letter of the axis name set by the parameter "#1022 axname2" exist, the axis address key moves the cursor to the first found axis (in the order of 1st axis --> 2nd axis --> additional axis --> 2nd additional axis). When you want to move the cursor to a low priority axis, input the axis name to be measured and the measurement value (radius value) of the workpiece before moving the cursor to the target axis with [|←] or [→|] key.

Value "Z0." is displayed in the input column.

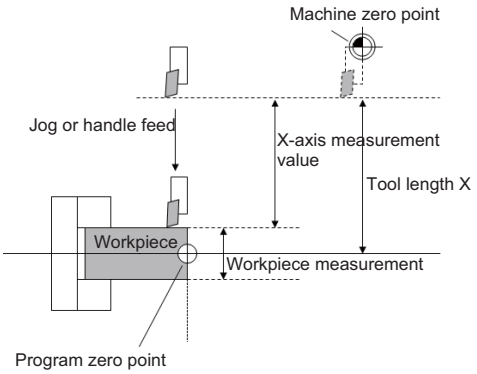
(7) Press the menu [Measure].

The measured value (Z axis machine position - Input value (0)) is set in the tool length data at the cursor position.
The input column is cleared.

<Note>

- For the axis to which the mirror image is turned ON, the measurement position considering the mirror state is displayed.

(8) Select the X axis, and manually move the tool to the work side face.



(9) Enter the name of axis to be measured, and the measurement value (diameter) of workpiece.

X20.

When you enter "X", the cursor is moved to the X-axis column.

<Note>

- If you input an axis name that does not exist, the cursor does not move.
- The second and subsequent characters are not considered to be the axis name and, therefore, the cursor does not move. (Example: If you enter "XZ", the cursor moves on the X axis, but the cursor does not move on the Z axis.)
  Also, if anything other than tool length data is displayed, the cursor does not move even when the first input value is the axis name.
- If multiple axes which have the same first letter of the axis name set in the parameter "#1022 axname2" exist, the axis address key moves the cursor to the first found axis (in the order of 1st axis --> 2nd axis --> additional axis --> 2nd additional axis). When you want to move the cursor to a low priority axis, input the axis name to be measured and the measurement value (radius value) of the workpiece before moving the cursor to the target axis with [|←] or [→|] key.

Value "X20." is displayed in the input column.

(10) Press the menu [Measure].

The measured value (X-axis machine position - Input value (20)) is set in the tool length data at the cursor position. The input column is cleared.

<Note>

- For the axis to which the mirror image is turned ON, the measurement position considering the mirror status is displayed.

Note

(1) If the parameter "#11017 T-ofs set at run" is set to "1", the tool compensation amount data during automatic operation or during pause of operation can be measured.

(2) This setting may be disabled due to the parameter "#8933 Disable lngth comp" or "8934 Disable wear comp" setting. For details, refer to "6.4.6 Disabling the Setting of Tool Compensation Amount".

(3) When entering data without decimal points, you can select the 1-mm (or 1-inch) data input or the input in the minimum input unit (same as "#1003 iunit") by designating the "#8119 Comp. unit switch" parameter.

<1-mm unit: "#8119 Comp. unit switch" is "0".>

Input	Result
1	1.000
100	100.000
10.	10.000
.012	0.012

<Minimum input unit (#1003 iunit B): "#8119 Comp. unit switch" is "1".>

Input	Result
1	0.001
100	0.100
10.	10.000
.012	0.012

Measuring the tool compensation amount of Z axis and X axis (diameter axis) with the parameter designation method (when #11087 = 1)

(1) Select the tool to be measured with manual numerical value command. (Refer to “9.6 Manual Numerical Value Command" for details.)

(Example) T1003

(2) Displays the tool compensation amount screen.

Menu [Offset]

→ The tool compensation amount screen is displayed. The tool compensation number of the selected tool is displayed at the top, and the cursor is located on the first line.

<Note>

- If the parameter "#8972 T code offset disp" is set to "0", the tool compensation number of selected tool is not displayed at the top. Locate the cursor onto the tool compensation number of the tool to be measured.

(3) Press the menu [Length data].

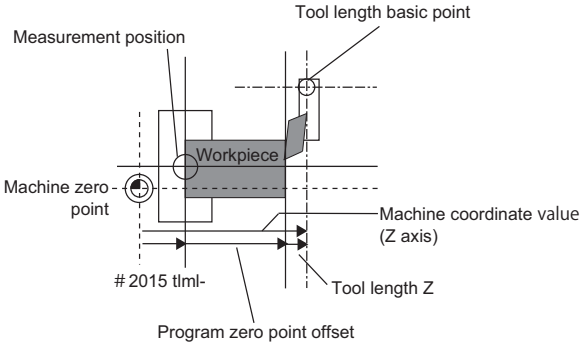
→ This switches to the tool length data display.

<Note>

- When the axis to apply the mirror image is to be measured, turn ON the mirror image.

(4) Select the "Manual handle feed" mode or "Jog feed" mode.

(5) Select the Z axis, and manually move the tool to the work side face.



(6) Enter the name of axis to be measured, and the measurement value (radius) of workpiece.

Z50.

When you enter "Z", the cursor is moved to the Z axis column.

<Note>

- If you input an axis name that does not exist, the cursor does not move.
- The second and subsequent characters are not considered to be the axis name and, therefore, the cursor does not move. (Example: If you enter "XZ", the cursor moves on the X axis, but the cursor does not move on the Z axis.)
  Also, if anything other than tool length data is displayed, the cursor does not move even when the first input value is the axis name.
- If multiple axes which have the same first letter of the axis name set by the parameter "#1022 axname2" exist, the axis address key moves the cursor to the first found axis (in the order of 1st axis --> 2nd axis --> additional axis --> 2nd additional axis). When you want to move the cursor to a low priority axis, input the axis name to be measured and the measurement value (radius value) of the workpiece before moving the cursor to the target axis with [|←] or [→|] key.

Value "Z50." is displayed in the input column.

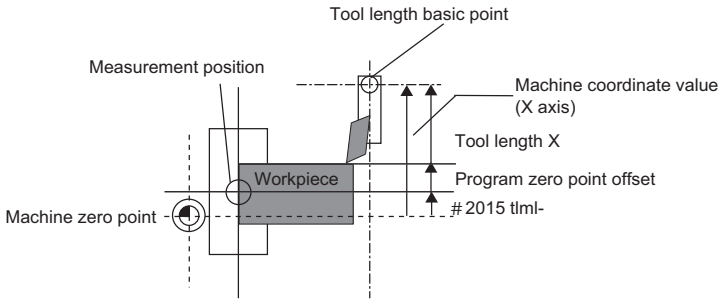
(7) Press the menu [Measure].

The measured value (Z axis machine position - measurement zero point (#2015 tlml-) - Input value (50)) is set in the tool length data at the cursor position.
The input column is cleared.

<Note>

- For the axis to which the mirror image is turned ON, the measurement position considering the mirror status is displayed.

(8) Select the X axis, and manually move the tool to the work side face.



(9) Enter the name of axis to be measured, and the measurement value (diameter) of workpiece.

X20.

When you enter "X", the cursor is moved to the X-axis column.

<Note>

- If you input an axis name that does not exist, the cursor does not move.
- The second and subsequent characters are not considered to be the axis name and, therefore, the cursor does not move. (Example: If you enter "XZ", the cursor moves on the X axis, but the cursor does not move on the Z axis.)
- If multiple axes which have the same first letter of the axis name set by the parameter "#1022 axname2" exist, the axis address key moves the cursor to the first found axis (in the order of 1st axis --> 2nd axis --> additional axis --> 2nd additional axis). When you want to move the cursor to a low priority axis, input the axis name to be measured and the measurement value (radius value) of the workpiece before moving the cursor to the target axis with [|←] or [→|] key.

Value "X20." is displayed in the input column.

(10) Press the menu [Measure].

The measured value (X axis machine position - measurement zero point (#2015 tlml-) - Input value (20)) is set in the tool length data at the cursor position.
The input column is cleared.

<Note>

- For the axis to which the mirror image is turned ON, the measurement position considering the mirror status is displayed.

Note

(1) If the parameter "#11017 T-ofs set at run" is set to "1", the tool compensation amount data during automatic operation or during pause of operation can be measured.

(2) This setting may be disabled due to the parameter "#8933 Disable lngth comp" or "8934 Disable wear comp" setting. For details, refer to "6.4.6 Disabling the Setting of Tool Compensation Amount".

(3) When entering data without decimal points, you can select the 1-mm (or 1-inch) data input or the input in the minimum input unit (same as "#1003 iunit") by designating the parameter "#8119 Comp. unit switch".

<1-mm unit: "#8119 Comp. unit switch" is "0".>

Input	Result
1	1.000
100	100.000
10.	10.000
.012	0.012

<Minimum input unit (#1003 iunit B): "#8119 Comp. unit switch" is "1".>

Input	Result
1	0.001
100	0.100
10.	10.000
.012	0.012

Measuring the tool compensation amount of Z axis and X axis (diameter axis) with the workpiece coordinate system offset designation method (when #11087 = 2)

(1) Select the tool to be measured with manual numerical value command. (Refer to “9.6 Manual Numerical Value Command" for details.)

(Example) T1003

(2) Displays the tool compensation amount screen.

Menu [Offset]

➡ The tool compensation amount screen is displayed. The tool compensation number of the selected tool is displayed at the top, and the cursor is located on the first line.

<Note>

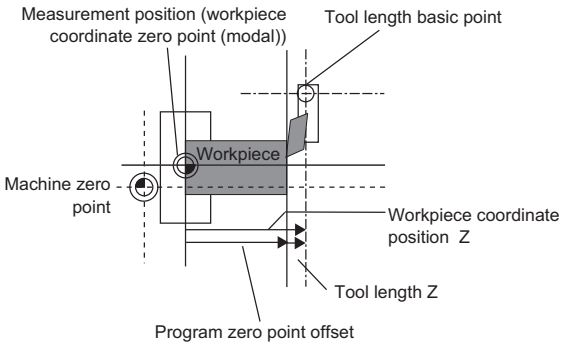
- If the parameter "#8972 T code offset disp" is set to "0", the tool compensation number of selected tool is not displayed at the top. Locate the cursor onto the tool compensation number of the tool to be measured.

(3) Press the menu [Tool length data].

➡ This switches to the tool length data display.

(4) Select the "Manual handle feed" mode or "Jog feed" mode.

(5) Select the Z axis, and manually move the tool to the work side face.



(6) Enter the name of axis to be measured, and the measurement value (radius) of workpiece.

Z50.

When you enter "Z", the cursor is moved to the Z axis column.

<Note>

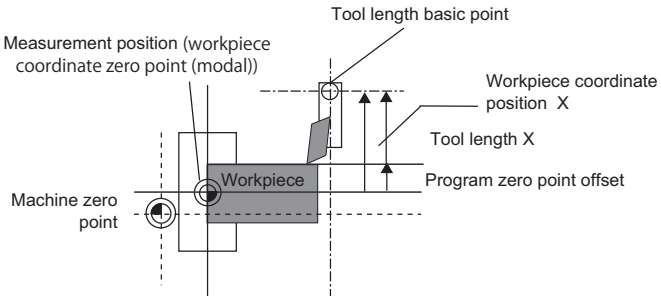
- If you input an axis name that does not exist, the cursor does not move.
- The second and subsequent characters are not considered to be the axis name and, therefore, the cursor does not move. (Example: If you enter "XZ", the cursor moves on the X axis, but the cursor does not move on the Z axis.)
  Also, if anything other than tool length data is displayed, the cursor does not move even when the first input value is the axis name.
- If multiple axes which have the same first letter of the axis name set by the parameter "#1022 axname2" exist, the axis address key moves the cursor to the first found axis (in the order of 1st axis --> 2nd axis --> additional axis --> 2nd additional axis). When you want to move the cursor to a low priority axis, input the axis name to be measured and the measurement value (radius value) of the workpiece before moving the cursor to the target axis with [|←] or [→|] key.

Value "Z50." is displayed in the input column.

(7) Press the menu [Measure].

The measured value (Z axis workpiece coordinate position - Input value (50)) is set in the tool length data at the cursor position.
The input column is cleared.

(8) Select the X axis, and manually move the tool to the work side face.



(9) Enter the name of axis to be measured, and the measurement value (diameter) of workpiece.

X20.

When you enter "X", the cursor is moved to the X-axis column.

<Note>

- If you input an axis name that does not exist, the cursor does not move.
- The second and subsequent characters are not considered to be the axis name and, therefore, the cursor does not move. (Example: If you enter "XZ", the cursor moves on the X axis, but the cursor does not move on the Z axis.)
  Also, if anything other than tool length data is displayed, the cursor does not move even when the first input value is the axis name.
- If multiple axes which have the same first letter of the axis name set by the parameter "#1022 axname2" exist, the axis address key moves the cursor to the first found axis (in the order of 1st axis --> 2nd axis --> additional axis --> 2nd additional axis). When you want to move the cursor to a low priority axis, input the axis name to be measured and the measurement value (radius value) of the workpiece before moving the cursor to the target axis with [|←] or [→|] key.

Value "X20." is displayed in the input column.

(10) Press the menu [Measure].

The measured value (X axis workpiece coordinate position - Input value (20)) is set in the tool length data at the cursor position.
The input column is cleared.

Note

(1) If the parameter "#11017 T-ofs set at run" is set to "1", the tool compensation amount data during automatic operation or during pause of operation can be measured.

(2) This setting may be disabled due to the parameter "#8933 Disable lngth comp" or "8934 Disable wear comp" setting. For details, refer to "6.4.6 Disabling the Setting of Tool Compensation Amount".

(3) When entering data without decimal points, you can select the 1-mm (or 1-inch) data input or the input in the minimum input unit (same as "#1003 iunit") by designating the "#8119 Comp. unit switch" parameter.

<1-mm unit: "#8119 Comp. unit switch" is "0".>

Input	Result
1	1.000
100	100.000
10.	10.000
.012	0.012

<Minimum input unit (#1003 iunit B): "#8119 Comp. unit switch" is "1".>

Input	Result
1	0.001
100	0.100
10.	10.000
.012	0.012

6.4.8 Moving to designated Compensation Number

There are two methods to switch from the designated compensation number display. You can switch the operation by using the "#8975 Change No. search" parameter.

(Example 1) Select compensation number 16. (If "#8975 Change No. search" is "0")

(1) Displays the tool compensation amount screen.

(2) Press the menu [Ofs No search].

(3) Enter the compensation number you wish to display, and press [INPUT].
16 [INPUT]
➡ The compensation number that you have set is displayed at the top, and the cursor is located on the first data. The highlighting of the menu [Ofs No. search] is canceled.

(Example 2) Select compensation number 16. (If "#8975 Change No. search" is "1")

(1) Displays the tool compensation amount screen.

(2) Enter the compensation number you wish to display.
16

(3) Press the menu [Ofs No search].
➡ The compensation number that you have set is displayed at the top, and the cursor is located on the first data.

Note

(1) If an compensation number exceeding the limit is designated, the operation message "Setting error" is displayed.

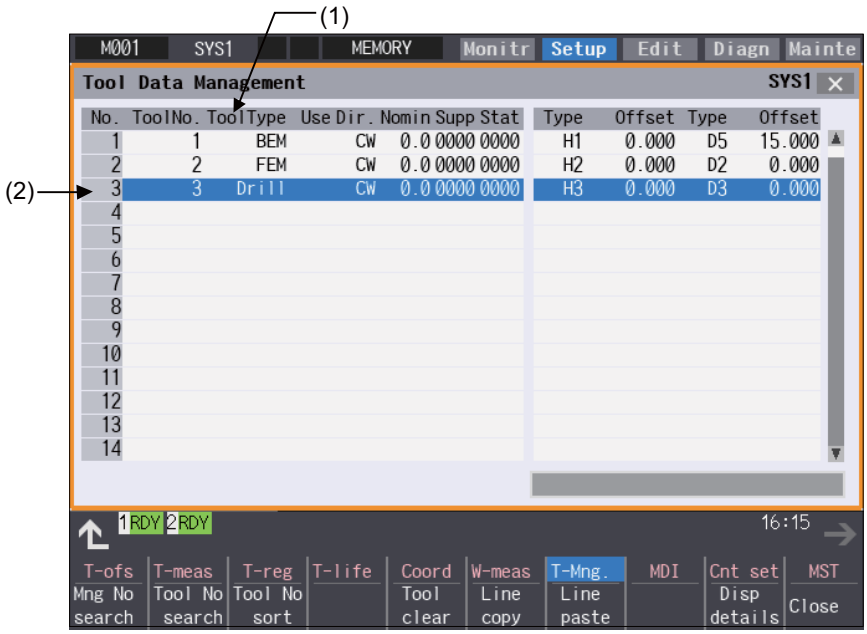
6.5 Setting the Tool Management Data (Tool Management Screen)

Setup – T-Mng. (M system)

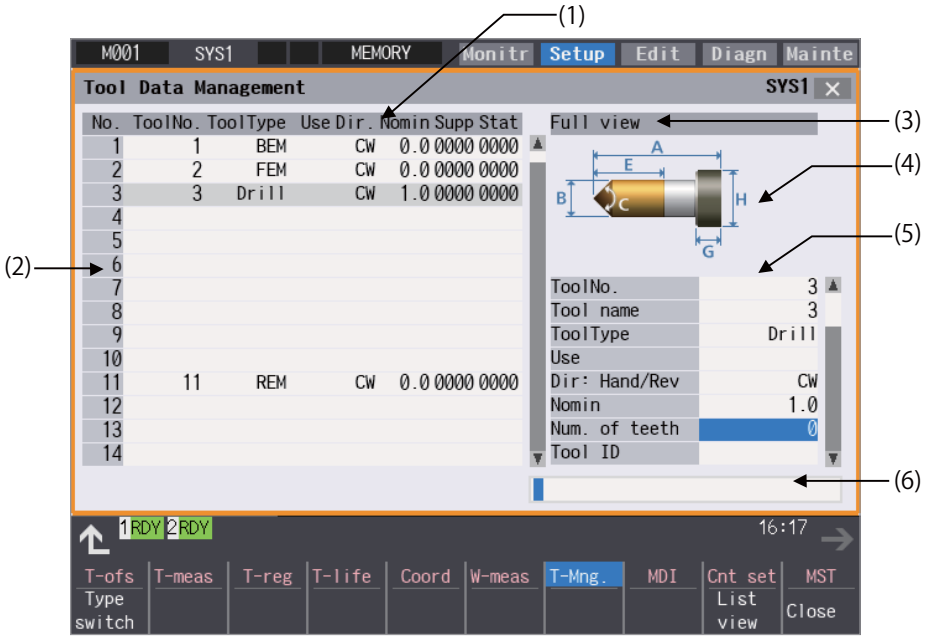
Setup – T-Mng. (L system)

Tool management data can be set and displayed on the tool management (T-Mng.) screen. These data are relevant to which displayed on the [T-ofs] or [T-life] screen, that can be set the tool information. Thus the mutually-referenced data can be set and displayed on this screen.

■List view



■Detailed display



Display items

Display items		Details
(1)	Tool management data list portion	The preset tool management data is listed. The tool management data list consists of the right side and left side parts. When you select the menu [Disp details], the right side of the tool management data list is changed to the detailed tool information view.
(2)	Data No.	Indicates the number of the tool management data.
(3)	Display data type	It indicates the item type that is displayed in the detail view of tool management data. The display data type can be "Full view" (*1), "Custom" (*2)(*3), "Basic info", "Shape info", "Cut cond.", "Offset", "T-life", or "Added info". After the power is turned ON again, the last display data type is maintained.
(4)	Tool shape drawing	It displays the tool shape of the tool management data at cursor position according to the tool type. The display and hiding are switched according to the display data type.
(5)	Tool management data detailed portion	Displays or sets the tool data that has been selected from the tool management data list. The display items are changed depending on the tool type and the display data type.
(6)	Input section	This displays the input data.

(*1) The display items differ depending on MTB settings.

(*2) The display data type "Custom" may not be displayed depending on MTB settings.

(*3) The type names and item list differ depending on MTB settings.

6.5.1 List of Display Items

○: Display/setting enabled ●: Display only -: Non-display

Display data			Tool management data list portion		Tool management data detailed portion							
			Left side	Right side	Full view	Custom (*9)	Basic info	Shape info	Cut cond.	Offset	T-life	Added info
Tool shape drawing			-	-	○	○	○	○	-	-	-	-
Tool management data	Basic info	Tool No.	○	-	○	○	○	○	○	○	○	○
		Tool name	-	-	○ (*10)	○ (*10)	○	-	-	-	-	-
		Tool type	○	-	○	○	○	○	○	○	○	○
		Use	○	-	○ (*10)	○ (*10)	○	-	-	-	-	-
		Dir: Hand/ Rev (*1)	○	-	○ (*10)	○ (*10)	○	-	-	-	-	-
		Nomin	○	-	○ (*10)	○ (*10)	○	-	-	-	-	-
		Num. of teeth	-	-	○ (*10)	○ (*10)	○	-	○ (*8)	-	-	-
		Tool ID	-	-	○ (*10)	○ (*10)	○	-	-	-	-	-
		Supp	○	-	○ (*10)	○ (*10)	-	-	-	-	-	-
		Stat	●	-	● (*10)	● (*10)	-	-	-	-	-	-
		Setting An-gle	-	-	○ (*10)	○ (*10)	-	○	-	-	-	-
		GangTool ofs Y	-	-	○ (*10)	○ (*10)	-	○	-	-	-	-
		Mount status	-	-	○ (*10)	○ (*10)	-	-	-	-	-	-
	Shape info (*2)		-	-	○ (*10)	○ (*10)	-	○	-	-	-	-
	Cut cond.	Spindle rpm S	-	-	○ (*10)	○ (*10)	-	-	○	-	-	-
		Feedrate F	-	-	○ (*10)	○ (*10)	-	-	○	-	-	-
		Material	-	-	○ (*10)	○ (*10)	-	-	○	-	-	-
		Coolant M code	-	-	○ (*10)	○ (*10)	-	-	○	-	-	-
	Added info	Custom 1 to 12 (*3)	-	-	○ (*4)(*10)	○ (*10)	-	-	-	-	-	○(*4)
		Custom 13 to 50 (*3)(*11)	-	-	○ (*4)(*10)	○ (*10)	-	-	-	-	-	○(*4)
	Life (*5)		-	-	-	○ (*10)	-	-	-	-	○	-
	Compensation		-	○ (*6)	-	○ (*10)	-	-	-	○ (*7)	-	-

(*1) When mill tools, lathing drill or lathing tap is selected, "Rev" is applied, and when lathe turning, groove or thread cut is selected, "Hand" is applied.

(*2) The display/setting contents differ depending on the tool types. Refer to "List of display items for shape info" in this section for details of setting items.

(*3) The display/setting format and setting rage of display item names, number of items and setting values depend on MTB specifications.

(*4) When the parameter "#1445 Tol-Custom-nondisp" is set to "1", "Added info" is not displayed.

(*5) The display/setting contents differ depending on the tool compensation type and tool life management type. Refer to "List of display items for life data" in this section for details of setting items.

(*6) The display contents differ depending on the tool compensation type and parameter "#8969 Tool offset type 1" and "#8970 Tool offset type 2". Refer to "List of display items for compensation data (List view)" in this section for details of setting items.

(*7) The display/setting contents differ depending on the tool compensation type. Refer to "List of display items for compensation data (Disp details)" in this section for details of setting items.

(*8) The item displays between "Material" and "Coolant M code".

(*9) The type name "Custom" differs depending on MTB specifications.

(*10) This may not be displayed depending on MTB specifications.

(*11) This is displayed when the parameter "#1766 cfgPR16/bit4" is set to "1".

List of display items for shape info

The display contents of shape info differ depending on the selected tools.

[Mill tools]

When the tool type is not specified, it is regarded as "Drill".

○: Display/setting enabled -: Not used

Item	Tool type							
	Not set	Ball end mill	Flat end mill	Drill	Radius end mill	Chamfer	Tap	Face mill
:A	Length							
:B	Diameter							
:C	-	-	-	Nose angle	Corner R	Tip angle	Pitch	Cutter L
:D	-	-	-	-	-	Tip dia.	Root dia.	Shank dia.
:E	-	Blade length	Blade length	Blade length	Blade length	Blade length	-	Blade length
:F	-	-	-	-	-	-	-	-
:G	Holder H							
:H	Holder D							
3D CHK Tool CR	3D CHK Tool CR							

Item	Tool type						
	Boring	Back boring	Tap - UN No.	Tap - UN Frac	Tap - Pipe Rc	Tap - Pipe G	Tap - Pipe Rp
:A	Length						
:B	Diameter						
:C	-	-	-	Denominator	Denominator	Denominator	Denominator
:D	-	-	-	Numerator	Numerator	Numerator	Numerator
:E	-	-	Threads/ inch	Threads/ inch	-	-	-
:F	-	-	Tap method	Tap method	Tap method	Tap method	Tap method
:G	Holder H						
:H	Holder D						
3D CHK Tool CR	3D CHK Tool CR						

[Turning tools]

○: Display/setting enabled -: Not used

Item	Tool type				
	Lathe turning	**Groove**	**Thread cut**	**Lathing drill**	**Lathing tap**
:A	Tool length A				
:B	Tool length B				
:C	Nose R	Nose R	-	Nose angle	Pitch
:D	Nose angle	Nose width	-	-	Root dia.
:E	Cutting edge angle	Maximum groove depth	-	Blade length	-
:F	Tool width	Tool width	Tool width	-	Tap method
:G	Holder H				
:H	Holder D				
3D CHK Tool CR	3D CHK Tool CR				

Item	Tool type				
	Turn-UN No.	**Turn-UN Frac**	**Turn-PipeRc**	**Turn-Pipe G**	**Turn-PipeRp**
:A	Length				
:B	Diameter				
:C	-	Denominator	Denominator	Denominator	Denominator
:D	-	Numerator	Numerator	Numerator	Numerator
:E	Threads/inch	Threads/inch	-	-	-
:F	Tap method	Tap method	Tap method	Tap method	Tap method
:G	Holder H				
:H	Holder D				
3D CHK Tool CR	3D CHK Tool CR				

List of display items of life data

○: Display/setting enabled ●: Display only -: Non-display

Display data (Life data)		M system		L system	
		Tool life management I/II	Tool life management III	Tool life management I	Tool life management II
Life	Group No.	○	-	-	○
	Status	○	○	-	○ (*2)
	Mthd	○	○	-	●
	H No.	○	○	-	-
	L compen	○	○	-	-
	D No.	○	○	-	-
	R compen	○	○	-	-
	Auxil.	○	○	-	-
	Ofset No	-	-	-	○
	Life (*1)	○	○	-	●
	Used (*1)	○	○	-	○
	Usage time (*1)	-	-	○	-
	Life time (*1)	-	-	○	-
	Usage count (*1)	-	-	○	-
	T life count (*1)	-	-	○	-
	Status A	-	-	○	-
	Status B	-	-	○	-

(*1) Setting values and units are displayed.

(*2) Status is displayed under "Used".

List of display items for compensation data (List view)

The parameter "#8969 Tool offset type 1" enables to switch the display items for 1st and 2nd column and the parameter "#8970 Tool offset type 2" enables to switch the display items for 3rd and 4th column on the right side of the list view.

[Tool compensation type I]

Tool compensation No. displays in the 1st column, tool compensation amount displays in the 2nd column and 3rd and 4th column are set to blank, irrespective of the value set by the parameter.

#8969 Tool offset type 1

Setting value	1st column (type)	2nd column (compensation amount)
0 to 11	H tool compensation No.	Length compensation

#8970 Tool offset type 2 —: Blank

Setting value	3rd column	4th column
0 to 11	-	-

[Tool compensation type II/Tool compensation type III (M system)]

#8969 Tool offset type 1

Setting value	1st column (type)	2nd column (compensation amount)
0, 5 to 11	H tool compensation No.	Length compensation
1	H tool compensation No.	Length compensation
2	D tool compensation No.	Radius compensation
3	+H tool compensation No.	Length wear
4	+D tool compensation No.	Radius wear

#8970 Tool offset type 2

Setting value	3rd column (type)	4th column (compensation amount)
0, 5 to 11	D tool compensation No.	Radius compensation
1	H tool compensation No.	Length compensation
2	D tool compensation No.	Radius compensation
3	+H tool compensation No.	Length wear
4	+D tool compensation No.	Radius wear

[Tool compensation type III (L system)]

#8969 Tool offset type 1

Setting value	1st column (type) (*1)	2nd column (compensation amount)
0	"1st axis name" tool compensation No.	1st axis tool length
1	"1st axis name" tool compensation No.	1st axis tool length
2	"2nd axis name" tool compensation No.	2nd axis tool length
3	"Additional axis name" tool compensation No.	Additional axis tool length
4	"1st axis name" tool compensation No.	1st axis wear
5	"+ 2nd axis name" tool compensation No.	2nd axis wear
6	"+ Additional axis name" tool compensation No.	Additional axis wear
7	R	Nose R
8	+R	R wear
9	P	Point
10	"2nd additional axis name" tool compensation No.	2nd additional axis tool length
11	"+ 2nd additional axis name" tool compensation No.	2nd additional axis wear

#8970 Tool offset type 2

Setting value	3rd column (type) (*1)	4th column (compensation amount)
0	"2nd axis name" tool compensation No.	2nd axis tool length
1	"1st axis name" tool compensation No.	1st axis tool length
2	"2nd axis name" tool compensation No.	2nd axis tool length
3	"Additional axis name" tool compensation No.	Additional axis tool length
4	"1st axis name" tool compensation No.	1st axis wear
5	"+ 2nd axis name" tool compensation No.	2nd axis wear
6	"+ Additional axis name" tool compensation No.	Additional axis wear
7	R	Nose R
8	+R	R wear
9	P	Point
10	"2nd additional axis name" tool compensation No.	2nd additional axis tool length
11	"+ 2nd additional axis name" tool compensation No.	2nd additional axis wear

(*1) Names set in "#1013 axname" corresponding to the axis are displayed in "*** axis name".

Note

(1) For the compensation amount, when "#1041 Initial inch" is se to "0" (metric system), the fraction part is fixed in 3 digits and when it is set to "1" (inch system), the fraction part is fixed in 4 digits, and the values after those digits are cut off.

List of display items for compensation data (Disp details)

○: Display/setting enabled -: Non-display

Display data (Compensation data)		Tool compensation type I	Tool compensation type II	Tool compensation type III (M system)	
				L system	M system
Compensation	H No.	○	○	-	-
	D No.	-	○	-	-
	Length	○	○	-	-
	L wear	-	○	-	-
	Radius (*3)	-	○	-	-
	R wear (*3)	-	○	-	-
	Tool L offset #	-	-	○	○
	Wear offset #	-	-	○	○
	Tool length "1st axis name" (*1)(*2)(*4)(*7)	-	-	○	-
	Tool length "2nd axis name" (*1)(*2)(*4)(*7)	-	-	○	-
	Tool length "Additional axis name" (*1)(*2)(*4)(*7)	-	-	○	-
	Tool length "2nd additional axis name" (*1)(*4)(*5)(*7)	-	-	○	-
	Tool length "Axis name set in "base_I"" (*1)(*6)(*7)	-	-	-	○
	Tool length "Axis name set in "base_J"" (*1)(*6)(*7)	-	-	-	○
	Tool length "Axis name set in "base_K"" (*1)(*6)(*7)	-	-	-	○
	Wear "1st axis name" (*1)(*2)(*4)(*7)	-	-	○	-
	Wear "2nd axis name" (*1)(*2)(*4)(*7)	-	-	○	-
	Wear "Additional axis name" (*1)(*2)(*4)(*7)	-	-	○	-
	Wear "2nd additional axis name" (*1)(*4)(*5)(*7)	-	-	○	-
	Wear "Axis name set in "base_I"" (*1)(*6)(*7)	-	-	-	○
	Wear "Axis name set in "base_J"" (*1)(*6)(*7)	-	-	-	○
	Wear "Axis name set in "base_K"" (*1)(*6)(*7)	-	-	-	○
	Nose R (*3)	-	-	○	○
	R wear (*3)	-	-	○	○
	Point	-	-	○	○
	Side-cut	-	-	○	-
	End-cut	-	-	○	-

(*1) Names set by the parameter "#1022 axname2" corresponding to the axis are displayed in "*** axis name".

(*2) If the number of axes in the part system is 2 or less, the name of non-existing axis is displayed according to the setting of the parameter"#1281 ext17/bit1" (Selection of additional tool offset axis) as follows.
0: The axis name is not displayed.
1: The axis name designated by "#1028 base_K" is displayed in the 2nd axis, and the axis name designated by "#1027 base_J" is displayed in the additional axis.

(*3) When the parameter "#8117 OFS Diam DESIGN" is set to "1", the symbol "Φ" is displayed on the right side of the item name.

(*4) When "#1019 dia" (Diameter specification axis) is set to "1" for the axis, the symbol "Φ" is displayed on the right side of the item name.

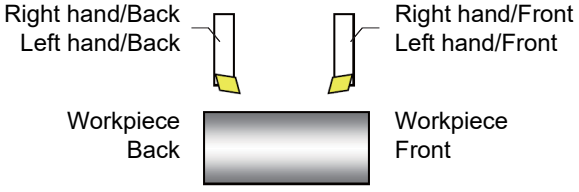
(*5) The axis that is set in "#12104 2nd add T-ofs set" is displayed in the 2nd additional axis name. If it does not set, the axis name is not displayed. Also, when "#12103 2nd add T-ofs ON" is set to "0", the line for the 2nd additional axis is not displayed.

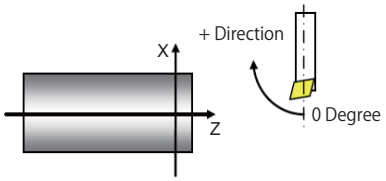
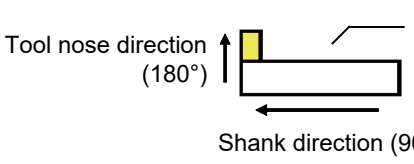
(*6) The 2nd axis names set by "#1022 axname2" are displayed for the axes set in the parameters "#1026 base_I", "#1027 base_J" and "#1028 base_K". However, if any axes designated by the parameter "#1026 base_I" to "#1028 base_K" does not exist, the axis name of the parameter "#1026 base_I" to "#1028 base_K" will be displayed instead.

(*7) The tool compensation data of an axis which has been set to "1" (hide) by the parameter "#12117 T-ofs hide axis" is not displayed.

6.5.2 Data Setting Range

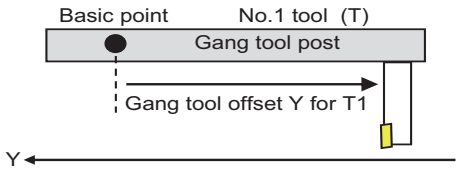
The value of each item is maintained even after the power is turned OFF and ON.
The setting value of each item retains when the tool type setting is changed.
Under setting range, the portions in parentheses are displayed in the "Tool Manage Data" section.

Setting item	Details	Setting range
Tool No.	Sets the tool No. A maximum of 8 digits of data can be input.	1 to 99999999
Tool name	Sets the tool name. A maximum of 8 alphanumeric characters and symbols can be set for the name. (*1)	A maximum of 8 characters
Type	This sets the tool type. The tool type will be "Drill" when no value is input.	1: Ball end mill (BEM) 2: Flat end mill (FEM) 3: Drill 4: Radius end mill (REM) 5: Chamfer 6: Tap 7: Face mill (Face) 8: Boring 9: Back boring (B boring) 10: Tap - Unified No. (TapUNNo.) 11: Tap - Unified denominator (TapUNFra) 12: Tap - pipe Rc (TapPipRc) 13: Tap - pipe G (TapPipeG) 14: Tap - pipe Rp (TapPipRp) 51: Lathe turning (Lathing) 52: Groove 53: Thread cutting (Thrding) 54: Lathing drill (L-drill) 55: Lathing tap (L-tap) 56: Turning - Unified No. (TrnUN-No.) 57: Turning - Unified denominator (TrnUN Fr) 58: Turning - Pipe Rc (TrnPipRc) 59: Turning - Pipe G (TrnPipeG) 60: Turning - Pipe Rp (TrnPipRp)
Use	This sets the usage of the tool.	0: No setting 1: Outer diameter 2: Inner diameter 3: End face - Front (End-F) 4: End face - Back (End-B)
Dir: Hand/Rev	This sets the tool hand and spindle rotation direction. The CW and CCW settings for the tool hands refer to the spindle rotation direction. The items to be set vary depending on the tool type. <Tools treated as the spindle rotation direction> Mill tools, lathing drills, lathing taps <Tools treated as handed/tool orientation/spindle rotation direction> Lathes, Grooves, Thread cutting For turning tool or grooving tool, the tool direction will change depending on the tool hand setting. <Example of turning tool>  	<Spindle rotation direction> 0: CW 1: CCW <Handed/tool orientation/spindle rotation direction> 0: Right/Front/CW (R/F) 1: Left/Front/CW (L/F) 2: Right/Back/CW (R/B) 3: Left/Back/CW (L/B) 4: Right/Front/CCW (R/F) 5: Left/Front/CCW (L/F) 6: Right/Back/CCW (R/B) 7: Left/Back/CCW (L/B)

Setting item	Details	Setting range
Nomin	Sets the nominal of the tools. When milling tools are selected in types, "nominal diameter" can be set and when turning tools are selected, "nominal" can be set. Settings will be different when lathing tap tools, turning-UN No. tools, tap tools, and tap-UN No. tools are selected in types.	Setting range varies depending on the parameter "#1041 Initial inch". mm: 0.0 to 999.9 Inches: 0.00 to 99.99 <Lathing tap tools, turning-UN No. tools, tap tools, and tap-UN No. tools> Lathing taps/taps: 1.0 to 99.9 Turning-UN No./Tap-UN No.: 0 to 99
Num. of teeth	Sets the number of teeth of the tools with 1 digit. When lathe turning, groove and thread cut tools are selected, it is grayed out and cannot be set.	0 to 9
Tool ID	Sets the information arbitrarily to identify the tools such as tool model No. described in a catalog. A maximum of 8 alphanumeric characters and symbols can be set for the ID. (*1)	A maximum of 8 characters
Supp	Sets the supplementary information of the tools. The display/setting values depend on MTB specifications.	0x0000 to 0xFFFF
Stat	Displays the status of the tools. It cannot be set and only reference is available. The displayed values depend on MTB specifications.	-
Installation angle	Sets the tool installation angle. Specifies the angle, assuming the direction perpendicular to Z axis as 0 degrees with CW as positive (+).  (Setting example) Setting angle = 90 Setting angle = 180 Setting angle = 270  When there is a tool rotary axis, set the angle assuming that the angle of the axis is 0 degrees. If the directions of the tool nose and shank are different such as internal diameter machining tool, set the tool nose angle for "Setting Angle". <Example of grooving tool for front face machining> 	0 to 359.999°

Setting item	Details	Setting range
Length: A	Tool type: all tools Sets the tool length.	<Length> Setting range varies depending on the parameter "#1041 Initial inch". mm: 0.000 to 9999.999 Inches: 0.0000 to 999.9999 <Angle> 0.000 to 180.000° <Thread denominator> 0: 1/1 1: 1/2 2: 1/4 3: 1/8 4: 1/16 For Turning - UN Frac/Tap - UN Frac 0 to 4 For Turning - Pipe Rc/Turning - Pipe G/Turning - Pipe Rp/Tap - Pipe Rc/Tap - Pipe G/Tap - Pipe Rp: 1 to 3 <Thread denominator> For Turning - UN Frac/Tap - UN Frac 0 to 99 For Turning - Pipe Rc/Turning - Pipe G/Turning - Pipe Rp/Tap - UN Frac/Tap - Pipe Rc/Tap - Pipe G/Tap - Pipe Rp: 0 to 9 < Threads/inch> 1 to 99 <Tap method> 0: Asynchronous 1: Synchronous
Diameter: B	Tool type: Mill tool, Lathing drill, Lathing tap, Turn-UN No., Turn-UN Frac, Turn - Pipe Rc, Turn - Pipe G, Turn - Pipe Rp Sets the tool diameter. (*2)	
Length: B	Tool type: Lathe turning, groove, thread cut Sets the tool length.	
Nose angle: C	Tool type: Lathing drill, drill Sets the nose angle.	
Corner R: C	Tool type: Radius end mill Sets the corner rounding circular radius.	
Tip angle: C	Tool type: Chamfer Sets the tip angle.	
Pitch : C	Tool type: Lathing tap, tap Set the pitch.	
Cutter L: C	Tool type: Face mill Sets the cutter length.	
Nose R : C	Tool type: Lathe turning, groove Sets the nose R.	
Denominator: C	Tool type: Turn-UN Frac, Turn-Pipe Rc, Turn-Pipe G, Turn-Pipe Rp, Tap-UN Frac, Tap-Pipe Rc, Tap-Pipe G,Tap-Pipe Rp Sets the outer diameter for the machined thread.	
Tip dia.: D	Tool type: Chamfer Sets the tool tip diameter. (*2)	
Root dia.: D	Tool type: Lathing tap, tap Sets the root diameter.	
Shank dia.: D	Tool type: Face mill Sets the shank diameter. (*2)	

Setting item	Details	Setting range
Nose angle: D	Tool type: Lathe turning Sets the nose angle.	<Length> Setting range varies depending on the parameter "#1041 Initial inch". mm: 0.000 to 9999.999 Inches: 0.0000 to 999.9999 <Angle> 0.000 to 180.000° <Thread denominator> 0: 1/1 1: 1/2 2: 1/4 3: 1/8 4: 1/16 For Turning - UN Frac/Tap - UN Frac 0 to 4 For Turning - Pipe Rc/Turning - Pipe G/Turning - Pipe Rp/Tap - Pipe Rc/Tap - Pipe G/Tap - Pipe Rp: 1 to 3 <Thread denominator> For Turning - UN Frac/Tap - UN Frac 0 to 99 For Turning - Pipe Rc/Turning - Pipe G/Turning - Pipe Rp/Tap - UN Frac/Tap - Pipe Rc/Tap - Pipe G/Tap - Pipe Rp: 0 to 9 <Threads/inch> 1 to 99 <Tap method> 0: Asynchronous 1: Synchronous
Nose width: D	Tool type: Groove Sets the nose width.	
Numerator: D	Tool type: Turn-UN Frac, Turn-Pipe Rc, Turn-Pipe G, Turn-Pipe Rp, Tap-UN Frac, Tap-Pipe Rc, Tap-Pipe G,Tap-Pipe Rp Sets the outer diameter for the machined thread.	
Edge angle: E	Tool type: Lathe turning Sets the cutting edge angle.	
Max groove: E	Tool type: Groove Sets the groove depth.	
Threads/inch: E	Tool type: Turn-UN No., Turn-UN Frac, Tap-UN No., Tap-UN Frac Sets the number of screw threads per inch.	
Blade length: E	Tool type: Ball end mill, flat end mill, drill, radius end mill, chamfer, face mill, lathing drill Sets the blade length.	
Tool width: F	Tool type: Lathe turning, groove, thread cut Sets the tool width.	
Tap method: F	Tool type: Lathing tap, Turn-UN No., Turn-UN Frac, Turn-Pipe Rc, Turn-Pipe G, Turn-Pipe Rp, Tap, Tap-UN Frac, Tap-Pipe Rc, Tap-Pipe G, Tap-Pipe Rp Sets the tap synchrony.	
Holder D: H	Tool type: all tools Sets the holder diameter.	
3D CHK Tool CR	Sets the color to be used in 3D check screen. Tool type: all tools The tool type will be red when no value is input. On the 3D machining simulation screen, this sets the color for the tool tip.	1: Gray 2: Red 3: Yellow 4: Blue 5: Green 6: Light blue 7: Purple 8: Pink
Spindle rpm S	Sets the spindle rotation speed.	0 to 99999999
Feedrate F	Sets the tool feedrate.	Setting range varies depending on the parameter "#1041 Initial inch". mm: 0 to 1000000 Inches: 0 to 100000
Material	Sets the material of the tool. A maximum of 4 alphanumeric characters and symbols can be set for the name. (*1)	A maximum of 4 characters
Coolant M code	Sets the coolant M code.	0 to 99999999

Setting item	Details	Setting range
GangTool ofs Y	For a machine equipped with a gang tool post, specify the basic point of the gang tool offset Y, and input the offset in Y axis direction between the basic point and each tool. You can determine the basic point in any desired position. You can determine the basic point arbitrarily. (Setting example) 	Setting range varies depending on the parameter "#1041 Initial inch". mm: -9999.999 to 9999.999 Inches: -999.9999 to 999.9999
Mount status	Sets whether the tool is installed in a turret or magazine.	0: Not mounted 1: Mounted
Custom 1 to 12	Sets the custom information.	Display format and setting range vary depending on the MTB specifications.
Group No.	Sets the group No. When the group No. is cleared, the life management data is also cleared.	0 to 99999999 Group No. is cleared when it is set to "0".
Status	Sets the status. (*3)	Setting range varies depending on the tool life management method. For details, refer to "6.6 Tool Life Management".
Mthd	Sets the method. (*4)(*5)	
H No.	Sets the H No. (*4)	
L compen	Sets the L offset. (*4)	
D No.	Sets the D No. (*4)	
R compen	Sets the R offset.(*4)	
Aux.	Sets the auxiliary data. (*4)	
Ofset No	The compensation No. is set in this column.	
Life	Sets the life. (*4)(*5)	
Used	Sets the used. (*3)	
Usage time	Sets the maximum life time.	
Life time	Sets the usage life time.	
Usage count	Sets the maximum life count.	
T life count	Sets the usage life count.	
Status A	Sets the status A.	
Status B	Sets the status B.	

Setting item	Details	Setting range
H No.	Sets the H No.	Setting range varies depending on the tool compensation type. For details, refer to "6.4 Setting Tool Compensation Amount (Tool Offset Screen)".
D No.	Sets the D No.	
Length	Sets the tool length compensation amount.	
L wear	Sets the compensation amount of tool length wear.	
Radius	Sets the tool radius compensation amount.	
R wear	Sets the tool radius wear compensation amount.	
Tool L offset #	Set the tool L offset No.	
Wear offset #	Set the wear offset No.	
Tool length data	<L system> Sets the tool length compensation amount of 1st axis, 2nd axis, additional axis, and 2nd additional axis. (*6) <M system> Sets the tool length compensation amount of the axes designated by the parameters "#1026 base_I", "#1027 base_J" and "#1028 base_K".	
Wear data	<L system> Sets the wear compensation amount of 1st axis, 2nd axis, additional axis, and 2nd additional axis. (*6) <M system> Sets the wear compensation of the axes designated by the parameters "#1026 base_I", "#1027 base_J" and "#1028 base_K".	
Nose R	Sets the nose R.	
R wear	Sets the R-wear.	
Point	Sets the point.	
Side-cut	Sets the side-cutting-edge angle. (*7)	
End-cut	Sets the end-cutting-edge angle. (*7)	

(*1) The symbols "/" (Slash), "\" (Backslash), and """ (Double quotes) cannot be used for names or IDs. Also, "=" (Equals) cannot be used as the first character.

(*2) When the parameter "#8968 Tool shape radius" is set to "0", input the value of diameter. When the parameter is set to "1", input the value of radius.

(*3) For tool life management I/II for the M system and tool life management II for the L system, this cannot be set if no group No. is set.

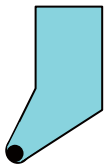
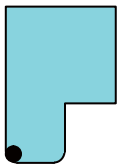
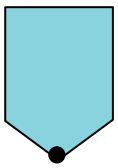
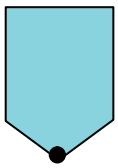
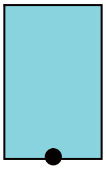
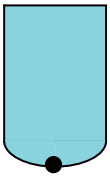
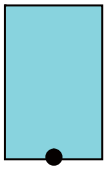
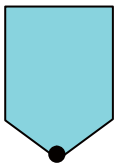
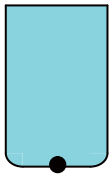
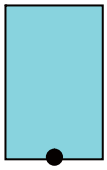
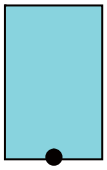
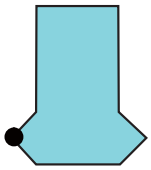
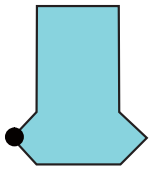
(*4) For tool life management I/II for the M system, this cannot be set if group No. is not set.

(*5) It cannot be set with tool life management II for L system. Only reference is available.

(*6) The axis names set by the parameter "#1022 axname2" are displayed for 1st axis, 2nd axis, additional axis, and 2nd additional axis.

(*7) It cannot be set for M system.

Refer to the figures below for the graphic and the nose position of each tool.

Lathe turning	Groove	Thread cut	Lathing drill	Lathing tap
				
Ball end mill	Flat end mill	Drill	Radius end mill	Chamfer
				
Tap	Face mill	Boring	Back boring	
				

● Indicates the tool nose position.

Note

(1) If a value outside of the setting range is entered, an error occurs, and the value cannot be set.

(2) When the sorting of tool data is in progress, the operation message "Sorting of tool data is in progress" appears, and the data cannot be set.

(3) Operations using the menu [Line paste] and [Tool clear], and changing each setting value cannot be performed during automatic operation or operation pause. However, setting value for some items can be changed depending on the setting of the parameter "#11017 T-ofs set at run" (Tool compensation amount setting during automatic operation enabled). Refer to the following table for details.

×: Cannot be changed Δ: Can be changed when #11017 is "1" -: Not managed

Display data			**Common**
Tool management data	Basic info	Tool No.	×

Display data			**M system**		**L system**	
			Tool life management I/II	**Tool life management III**	**Tool life management I**	**Tool life management II**
Life data	Life	Group No.	×	-	-	×
		Method	Δ	Δ	-	×
		Life	Δ	Δ	-	×
		Used	Δ	Δ	-	×

Display data			**Tool compensation type**		
			I	**II**	**III**
Compensation data	Compensation	H No.	×	×	-
		D No.	-	×	-
		Tool L offset num	-	-	×
		Wear offset #	-	-	×

(4) When the parameter "#1760 cfgPR10/bit6" (Wear compensation amount edit in data protection ON) is set to "1", wear compensation amount can be set regardless of whether the data protection key is ON or OFF.

6.5.3 Menu Configuration

Menus (List view)

Menus	Details
Mng No search	Displays the tool management data with the designated tool management number at top.
Tool No search	Displays the tool management data with the designated tool number at top.
Tool No sort	Sorts the tool No. in ascending order.
Tool clear	Clears the tool management data from the cursor line. Setting method: First data number to be cleared/last data number (Example) 10/30: The line data of data numbers 10 to 30 is cleared. 10/E: All line data that has data number 10 or less is cleared. If you press the [INPUT] key without entering the data number, data at the current cursor line is cleared.
Line copy	Copies a single line of data locating at the cursor position.
Line paste	Writes the copied data (single line data) in the cursor line.
Disp details	Switches to the detail view of tool management data.
Close	Returns from the tool management screen to the setup screen.

Menus (when the cursor is at "ToolNo." of [Disp details])

Menus	Details
Type switch	Switch the display data type of the detail view of tool management data by pressing the button.
Tool clear	Clear the tool management data of the currently selected tool number.
List view	Switch to the tool management data list.
Close	Returns from the tool management screen to the setup screen.

Menus (when the cursor is at "ToolType" of [Disp details])

Note

(1) For the L system, the menu buttons are displayed in a different order, sequentially from [Lathe turning].

Menus	Details
Type switch	Switch the display data type of the detail view of tool management data by pressing the button.
Ball endmill	Select the [Ball endmill]. It is also selected if the value "1" is entered.
Flat endmill	Select the [Flat endmill]. It is also selected if the value "2" is entered.
Drill	Select the [Drill]. It is also selected if the value "3" is entered.
Radius endmill	Select the [Radius end mill]. It is also selected if the value "4" is entered.
Chamfer	Select the [Chamfer]. It is also selected if the value "5" is entered.
Tap	Select the [Tap]. It is also selected if the value "6" is entered.
Face mill	Select the [Face mill]. It is also selected if the value "7" is entered.
Close	Returns from the tool management screen to the setup screen.
Boring	Select the [Boring]. It is also selected if the value "8" is entered.
Back Boring	Select the [Back Boring]. It is also selected if the value "9" is entered.
Tap - UN No.	Select the [Tap-UN No.]. It is also selected if the value "10" is entered.
Tap - UN Frac	Select the [Tap-UN Frac]. It is also selected if the value "11" is entered.
Tap - Pipe Rc	Select the [Tap-Pipe Rc] ("tapered thread" on the inner thread side). It is also selected if the value "12" is entered.

Menus	Details
Tap - Pipe G	Select the [Tap-Pipe G]. It is also selected if the value "13" is entered.
Tap - Pipe Rp	Select the [Tap-Pipe Rp] ("straight thread" on the inner thread side). It is also selected if the value "14" is entered.
Lathe turning	Select the [Lathing turning]. It is also selected if the value "51" is entered.
Groove	Select the [Groove]. It is also selected if the value "52" is entered.
Thread cut	Select the [Thread cut]. It is also selected if the value "53" is entered.
Lathing drill	Select the [Lathing drill]. It is also selected if the value "54" is entered.
Lathing tap	Select the [L-tap]. It is also selected if the value "55" is entered.
Turn - UN No.	Select the [Turn-UN No.]. It is also selected if the value "56" is entered.
Turn - UN Frac	Select the [Turn-UN Frac]. It is also selected if the value "57" is entered.
Turn - Pipe Rc	Select the [Turn-Pipe Rc] ("tapered thread" on the inner thread side). It is also selected if the value "58" is entered.
Turn - Pipe G	Select the [Turn-Pipe G]. It is also selected if the value "59" is entered.
Turn - Pipe Rp	Select the [Turn-Pipe Rp] ("straight thread" on the inner thread side). It is also selected if the value "60" is entered.

Menus (when the cursor is at "3D CHK Tool CR" of [Disp details])

Menus	Details
Type switch	Switch the display data type of the detail view of tool management data by pressing the button.
Gray	Select the gray color. It is also selected if the value "1" is entered.
Red	Select the red color. It is also selected if the value "2" is entered.
Yellow	Select the yellow color. It is also selected if the value "3" is entered.
Blue	Select the blue color. It is also selected if the value "4" is entered.
Green	Select the green color. It is also selected if the value "5" is entered.
Light blue	Select the aqua color. It is also selected if the value "6" is entered.
Purple	Select the purple color. It is also selected if the value "7" is entered.

Menus	Details
Pink	Select the pink color. It is also selected if the value "8" is entered.
Close	Returns from the tool management screen to the setup screen.

Menus (when the cursor is at "Use" of [Disp details])

Menus	Details
Type switch	Switch the display data type of the detail view of tool management data by pressing the button.
Leave blank	Select the [Leave blank]. This column is set to blank. It is also selected if the value "0" is entered.
Outer dia.	Select the [Outer dia.]. It is also selected if the value "1" is entered.
Inner dia.	Select the [Inner dia.]. It is also selected if the value "2" is entered.
EndFace Front	Select the [EndFaceFront]. It is also selected if the value "3" is entered.
EndFace Back	Select the [EndFaceBack]. It is also selected if the value "4" is entered.
List view	Switch to the tool management data list.
Close	Returns from the tool management screen to the setup screen.

Menus (when the cursor is at "Dir: Hand/Rev" of [Disp details] and mill tool, lathing drill, or lathing tap is selected)

Menus	Details
Type switch	Switch the display data type of the detail view of tool management data by pressing the button.
CW	Select the [CW]. It is also selected if the value "0" is entered.
CCW	Select the [CCW]. It is also selected if the value "1" is entered.
List view	Switch to the tool management data list.
Close	Returns from the tool management screen to the setup screen.

Menus (when the cursor is at "Dir: Hand/Rev" of [Disp details] and lathe, groove, or thread cutting is selected)

Menus	Details
Type switch	Switch the display data type of the detail view of tool management data by pressing the button.
R/Front /CW	Select the [R/Front/CW]. It is also selected if the value "0" is entered.
L/Front /CW	Select the [L/Front/CW]. It is also selected if the value "1" is entered.
R/Back /CW	Select the [R/Back/CW]. It is also selected if the value "2" is entered.
L/Back /CW	Select the [L/Back/CW]. It is also selected if the value "3" is entered.
R/Front /CCW	Select the [R/Front/CCW]. It is also selected if the value "4" is entered.
L/Front /CCW	Select the [L/Front/CCW]. It is also selected if the value "5" is entered.
R/Back /CCW	Select the [R/Back/CCW]. It is also selected if the value "6" is entered.
L/Back /CCW	Select the [L/Back/CCW]. It is also selected if the value "7" is entered.
List view	Switch to the tool management data list.
Close	Returns from the tool management screen to the setup screen.

Menus (List Menus (when none of the above conditions is met))

Menus	Details
Type switch	Switch the display data type of the detail view of tool management data by pressing the button.
List view	Switch to the tool management data list.
Close	Returns from the tool management screen to the setup screen.

Menus (when the cursor is at "Denominator" of [Disp details])

Menus	Details
Type switch	Switch the display data type of the detail view of tool management data by pressing the button.
1/1	Select [1/1]. It is also selected if the value "0" is entered. Only available if tool type is [Turn-UN Frac] or [Tap-UN Frac].
1/2	Select [1/2]. It is also selected if the value "1" is entered.
1/4	Select [1/4]. It is also selected if the value "2" is entered.
1/8	Select [1/8]. It is also selected if the value "3" is entered.
1/16	Select [1/16]. It is also selected if the value "4" is entered. Only available if tool type is [Turn-UN Frac] or [Tap-UN Frac].
List view	Switch to the tool management data list.
Close	Returns from the tool management screen to the setup screen.

Menus (when the cursor is at "Tap method" of [Disp details])

Menus	Details
Type switch	Switch the display data type of the detail view of tool management data by pressing the button.
Non-synch	Select [Non- synch]. It is also selected if the value "0" is entered.
Synch	Select [Synch]. It is also selected if the value "1" is entered.
List view	Switch to the tool management data list.
Close	Returns from the tool management screen to the setup screen.

6.5.4 Registering New Tool Management Data

Operation method

(1) Display the tool management screen. → The tool management data list is displayed.

If the details of tool management data is already displayed, press the menu [List view] and switch to the display of tool management data list.

(2) Use the [↑] or [↓] key to select the data number, which the tool No. is not specified yet.

(3) Press the [INPUT] key.

(4) Input the tool number.
100 [INPUT] → The tool management data of tool number 100 is registered in the selected data number, and the cursor is shifted to the next lower position.

6.5.5 Switching the Item Type of the Tool Management Data Detailed Portion

Operation method

(1) Display the tool management screen. → The tool management data list is displayed.

(2) Use the [↑] or [↓] key to select the data number to be displayed its details.

(3) Press the [INPUT] key.

(4) Press the menu [Type switch]. → Each time you press the menu, the item type shown in the detailed view is switched in the following sequence.
"Full view" -> "Custom" (*1) -> "Basic info" -> "Shape info" -> "Cut cond." -> "Offset" -> "T-life" -> "Added info" (*2) -> "Full view"
The cursor is moved to the top of the tool management data detail view. The selected type is maintained even after the power is turned ON again.

(*1) The type "Custom" is skipped if there is no setting by MTB.

(*2) If the machine parameter "#1445 Tol-Custom-nondisp" (Hide added info on tool management screen) is set to "1", the "Added info" type is skipped and it is switched to the next type.

6.5.6 Displaying Data by Designating Tool Management Data No.

There are two methods to switch the display from the designated tool management data number. You can switch the operation by using the "#8975 No. search process" parameter.

Operation method

(Example 1) Select data number 12. ("#8975 No. search process" is "0")

(1) Display the tool management screen. → The tool management data list is displayed.

(2) Press the menu [Mng No search].

(3) Enter the tool management data number to display, and press the [INPUT] key.
12 [INPUT]

(Example 2) Select data number 12. ("#8975 No. search process" is "1")

(1) Display the tool management screen. → The tool management data list is displayed.

(2) Enter the tool management data number to be displayed.
12

(3) Press the menu [Mng No search]. → The specified data number is displayed at the top, and the cursor is located onto the first data.

Note

(1) If a data number exceeding the limit is designated, the operation message "Setting error" is displayed.

6.5.7 Displaying Data by Designating Tool No.

Operation method ("#8975 No. search process" is "0")

The designated tool number is searched from the tool management data under the cursor position, and the tool number is displayed at the top when it is found. If the designated tool number is not found until the end of the tool management data, the search is performed from the head.

(Example) Search data number 12.

(1) Display the tool management screen. → The tool management data list is displayed.

(2) Press the menu [Tool No. search].

(3) Enter the tool management data number to display, and press the [INPUT] key.
12 [INPUT]
→ After the search is completed, the operation message "Search complete" appears.
The input column is cleared.
The designated tool number is searched from the tool management data under the cursor position, and first-found tool management data is displayed at the top.
The cursor is shifted to the head line of the display.
The highlight of the menu [Tool No. search] returns to normal.

Note

(1) If the tool number cannot be found, the message "Designated tool number not found" is displayed, and the input section is not cleared.

Operation method ("#8975 No. search process" is "1")

(Example) Search tool number 12.

The designated tool number is searched from the tool management data under the cursor position, and the tool number is displayed at the top when it is found. If the designated tool number is not found until the end of the tool management data, the search is performed from the head.
When you press the menu [Tool No. search] again after the search is completed, the search restart for the previously searched tool No.
The previously searched tool number is maintained until the tool management screen is closed, or the display is switched to the detailed view.

(1) Display the tool management screen. → The tool management data list is displayed.

(2) Enter the tool number you wish to display.
12

(3) Press the menu [Tool No. search].
→ After the search is completed, the operation message "Search complete" appears.
The input column is cleared.
The designated tool number is searched from the tool management data under the cursor position, and first-found tool management data is displayed at the top.
The cursor is shifted to the head line of the display.

(4) Press the menu [Tool No. search] again.

The search restart for the previously searched tool No.
After the search is completed, the operation message "Search complete" appears.
The input column is cleared.
The designated tool number is searched from the tool management data under the cursor position, and first-found tool management data is displayed at the top.
The cursor is shifted to the head line of the display.

Note

(1) If the tool number cannot be found, the message "Designated tool number not found" is displayed, and the input section is not cleared.

6.5.8 Sorting the Data by Tool No.

Sort the tool management data in ascending order of tool numbers.

Operation method

(1) Display the tool management screen. → The tool management data list is displayed.

(2) Press the menu [Tool No sort].

(3) Press the [Y] or [INPUT] key. →

The operation message "Sorting of tool data is in progress" flashes during tool data sorting.
The operation message "Sorting of tool data is complete" is displayed after the tool data has been sorted.
The tool management data is sorted in ascending order of tool numbers, and they are displayed from the beginning. The cursor moves to the top line.
The highlight of the menu [Tool No. sort] returns to normal.

Note

(1) During data sorting in ascending order of tool numbers, you cannot stop this sorting on the way.
(2) Once you have sorted the data in ascending order of tool numbers, you cannot return them to the previous sort order.
(3) If the same tool number is found multiple times, data is sorted in the order in which they are found from the top.
(4) If any line without tool number is found while the tool data is being sorted, that line is skipped to display.
(5) The tool data is sorted for the currently displayed part system.
(6) When the tool data is being sorted, screen operation is disabled. Operate the screen after the sorting is completed.
(7) When the tool data is being sorted, the screen cannot transit by direct screen selection.
(8) When the tool data is being sorted, the cycle start is disabled. If the the cycle start is performed, the error message "Tool data sorting in progress" appears.
(9) If the CNC power is ON while the tool data is being sorted, the tool management data may be corrupted. The data must be backed up in advance when the tool data is to be sorted.

6.5.9 Deleting the Tool Management Data

Deleting the tool management data in [List view]

■Method 1: Delete the tool management data of the cursor line.

(1) Display the tool management screen. → The tool management data list is displayed.

(2) Use the [↑] or [↓] key to move the cursor onto the line to be deleted.

(3) Delete the selected data.
Menu [Tool clear] [INPUT]

(4) Press the [Y] or [INPUT] key.
To cancel the erasing, press a key other than [Y] or [INPUT]. → Press any key except [Y] and [INPUT] keys to cancel.
Once deleted, the data is empty.

■Method 2: Delete the tool management data of the designated line.

(1) Display the tool management screen. → The tool management data list is displayed.

If the details of tool management data is displayed, press the menu [List view] and switch to the display of tool management data list.

(2) Press the menu [Tool clear].

(3) Specify the line clear range, by separating the first and last data numbers by a slash (/) from each other, and press the [INPUT] key.
1/5 [INPUT]
To delete data down to the last line, set "E".
(Example) 5/E:

(4) Press the [Y] or [INPUT] key.
To cancel the erasing, press a key other than [Y] or [INPUT]. → The tool management data, that corresponds to the designated data number, is cleared and highlight of the menu display is released.
Data of the cleared range is empty.

Note

(1) If a value outside of the clear range is specified, the operation message "Setting error" is displayed.

Deleting the tool management data in [Disp details]

(1) Display the tool management screen. → The tool management data list is displayed.

If the details of tool management data is displayed, press the menu [Disp details] and switch to the display of tool management data list.

(2) Use the [↑] or [↓] key to move the cursor onto the tool number.

(3) Delete the selected data.
Menu [Tool clear]

The menu [Tool clear] is displayed only when the cursor is located on the tool number.

(4) Press the [Y] or [INPUT] key.
To cancel the erasing, press a key other than [Y] or [INPUT]. → The tool management data, that is displayed in the detailed view of tool management data, is erased.

Note

(1) If you wish to change the tool data to be deleted, locate the cursor onto the tool management data list and select a tool data again.

6.5.10 Copying and Pasting the Tool Management Data

Operation method

(1) Display the tool management screen. → The tool management data list is displayed.

(2) Use the [↑] or [↓] key to move the cursor onto the line to be copied.

(3) Press the menu [Line copy].

(4) Use the [↑] or [↓] key to move the cursor onto the line to be pasted.

(5) Press the menu [Line paste]. → The copied data is written in the line at the cursor position.
The highlighted copy line is released.

Note

(1) Once data is copied, it is held until the next tool management data is copied or until the tool management screen is closed.

(2) For tool life management I, you cannot paste to the data that does not have the tool number. It causes a paste error.

(3) The information of "Mount status" is not pasted. The status retains "not mounted".

(4) The following information is not copied or pasted.

×: Not copied or pasted

Display data		Common
Tool shape drawing		×
Tool management data	Mount status	×

×: Not copied or pasted -: Not managed

Display dada (Life data)		M system		L system	
		Tool life management I/II	Tool life management III	Tool life management I	Tool life management II
Life	Status	×	×	-	×
	Method	×	×	-	×
	H No.	×	×	-	-
	L offset	×	×	-	-
	D No.	×	×	-	-
	R offset	×	×	-	-
	Aux.	×	×	-	-
	Offset num	-	-	-	×
	Life	×	×	-	×
	Used	×	×	-	×
	Operating time	-	-	×	-
	Life time	-	-	×	-
	T usage count	-	-	×	-
	T life count	-	-	×	-
	Status A	-	-	×	-
	Status B	-	-	×	-

×: Not copied or pasted -: Not managed

Display data (Compensation data)		Tool compensation type I	Tool compensation type II	Tool compensation type III	
				L system	M system
Compensation	Length	×	×	-	×
	L wear	-	×	-	×
	Radius	-	×	-	×
	R wear	-	×	-	×
	Tool L offset num	-	-	×	-
	Wear offset #	-	-	×	-
	Length 1st axis name	-	-	×	-
	Length 2nd axis name	-	-	×	-
	Length additional axis name	-	-	×	-
	Length 2nd additional axis name	-	-	×	-
	Wear 1st axis name	-	-	×	-
	Wear 2nd axis name	-	-	×	-
	Wear additional axis name	-	-	×	-
	Wear 2nd additional axis name	-	-	×	-
	Nose R	-	-	×	-
	R-wear	-	-	×	-
	Point	-	-	×	-
	Side-cut	-	-	×	-
	End-cut	-	-	×	-

6.5.11 Specifying Display Format and Setting Range of Additional Information

The display format and setting range of additional information can be changed by entering tool data file (TOOLALL.DAT) described in the specified format.

[Setting]

No.	Addresses	Details	Data range (Meaning)	Supplement
1	N	Custom data No.	1 to 12 (*1)	Custom 1 to 12
2	B	Number of bytes	0: 4 bytes 1: 1 byte 2: 2 bytes 4: 4 bytes	(*2)
3	K	Type	0: Signed decimal 1: Unsigned decimal 2: Hexadecimal 3: Bit 4: With decimal point (fixed number of digits) 5: With decimal point (linked with iunit and I_inch) 6: With decimal point (linked with iunit) 7: With decimal point (fixed number of digits, linked with I_inch) 8: Character string	♦Data with decimal point is valid only when it is 4 bytes and digits information is required. ♦Character string is valid only when it is 4 bytes.
4	D	Number of digits	Set a 2-digit decimal value. ♦Set the number of digits of the fraction part in ones place. ♦Set the number of digits of the integer part in tens place.	(*3)(*4)
5	L	Maximum value	Specify the configurable maximum value and minimum value. When a check is not necessary, set that minimum value is larger than or equal to maximum value. When "Type" is character string, the number of characters shall be 0 to 4. When the data with decimal point is selected for "Type", eliminate the decimal point from the value when iunit = B, and I_inch = 0 (mm). (Example 1) When the value is in the range of 0.000 to 99.999 Minimum value: 0, Maximum value: 99999 (Example 2) When the value is in the range of -99.999 to 10.000 Minimum value: -99999, Maximum value: 10000	
6	S	Minimum value		

(*1) When the parameter "#1766 cfgPR16/bit4" is "1", the setting range is "1 to 50".

(*2) When No. 3 "Type" is set to "3", the number of bytes is fixed to 1 byte. When "Number of bytes" is set to a value other than 1 byte, the set value is ignored and treated as 1 byte.

(*3) Set the number of digits according to the setting of No. 3 "Type". Set it so that the total number of digits of integer part and fraction part is 9 or less. When it is 10 or more, the correct value is not stored.

♦When "Type" is set to "0", "1", "2", "3", or "8"

Set "Number of digits" to "0". When a value other than "0" is set, the set value is ignored and treated as "0".

♦When "Type" is set to "4" or "7"

(Example) Set the number of digits as follows when the fraction part is 5 digits.

D45 (integer part 4 digits + fraction part 5 digits)

♦When "Type" is set to "5" or "6"

(Example) When the parameter "#1003 iunit" is set to "B" (0.001mm), the fraction part is 3 digits. Set the number of digits as follows.

D63 (integer part 6 digits + fraction part 3 digits)

(*4) When No. 3 "Type" is set to "5" or "7" and the parameter "#1041 I_inch" is set to "1" (Inch system), 1 digit is added to the fraction part set in No. 4 "Number of digits". When the input value is smaller than the set number of digits of the fraction part + 1, the custom data is displayed with "0" added to the end on the tool management screen.

Note

(1) When the illegal value is set, it is handled as 4 bytes, signed decimal data without range check.

[Setting example]

N1B1K0D0L100S-100
N2B0K4D33L999999S-999999

6.6 Tool Life Management

Setup – 

(M system)

Setup – 

(L system)

This function manages the tool life by monitoring the cumulative cutting hours and the cumulative number of cuttings. The tool which reaches the end of its tool life can be exchanged at proper timing, this realizes uniform quality machining. Tool life information (the estimated tool life, the estimated number of cuttings, etc.) can be registered on the tool life management screen.

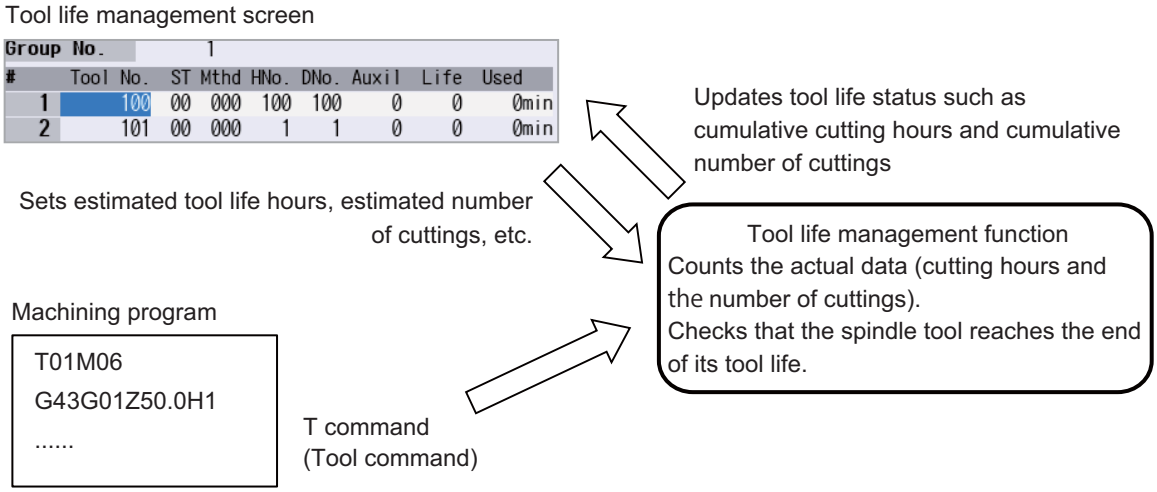
The tool life management function is enabled when the parameter "#1103 TLife" is set to "1"

There are some types of the tool life management method, two for M system and three for L system. It depends on the MTB specifications which type is to be selected. (The setting can be switched by the parameter "#1096 T_Ltyp")

Setting value of #1096	Machining center system (M system)	Lathe system (L system)
1	Tool life management I	Tool life management I
2	Tool life management II	Tool life management II
3	Tool life management III	(*1)

(*1) When "3" is set for L system, the tool life management I is applied.

For details of the respective management types, refer to "6.6.1 Tool Life Management (M System)" and "6.6.2 Tool Life Management (L System)".



6.6.1 Tool Life Management (M System)

Select a type of the tool life management and register the tool life data, and then issue the T command (tool command) to start the tool life management.

■Selecting the tool life management type

Features of the respective tool life managements are described in the table below.

Type	Feature	Group registration	Tool selection
I	The usage of a tool commanded by the machining program is monitored.	Available	Selected by the T command (tool command) directly
II	Command a group No./a tool No. to which the same type of the tool is registered by the machining program, and the corresponding tool is automatically selected form the group to monitor its usage. (*1)	Available	Selected from the group automatically (Automatic spare tool selection function)
III	The usage of a tool commanded by the machining program is monitored.	Not available (Group 1 only)	Selected by the T command (tool command) directly

(*1) The setting to handle the tool command (as the group No. or the tool No.) is specified by the parameter "#1104 T_Com2".
The method to select the tool from the group (in the order of the tool registration or of the length of the remaining tool life) is specified by the parameter "#1105 T_Sel2".

■Registering the tool life data

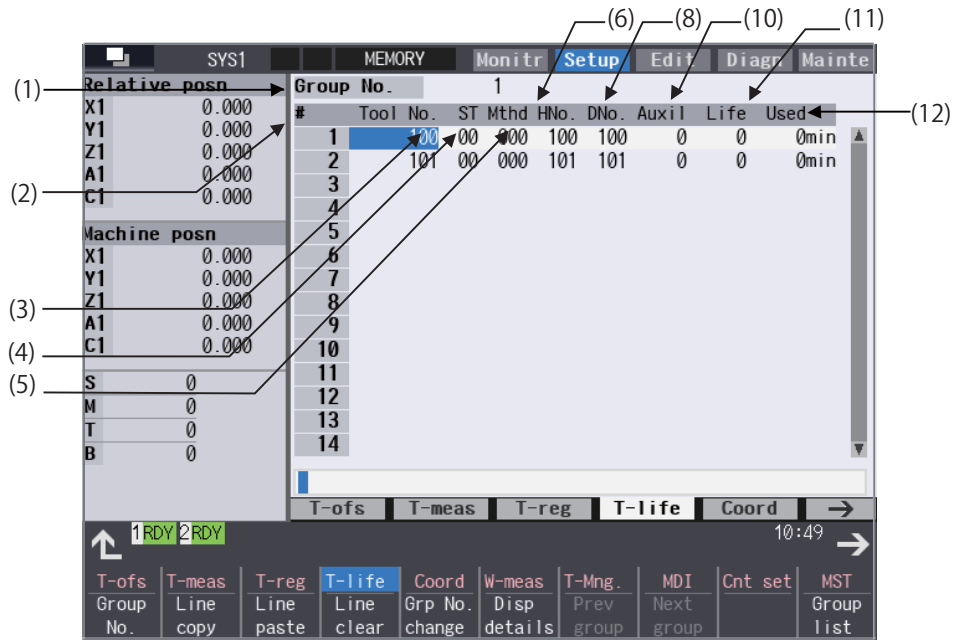
Register the tool life data which is to be used on the tool life management screen. The tool life data is set by the estimated tool life (hours) or by the estimated number of cuttings.

6.6.1.1 Registering Tool Life Management Data on the Group Unit Display Screen

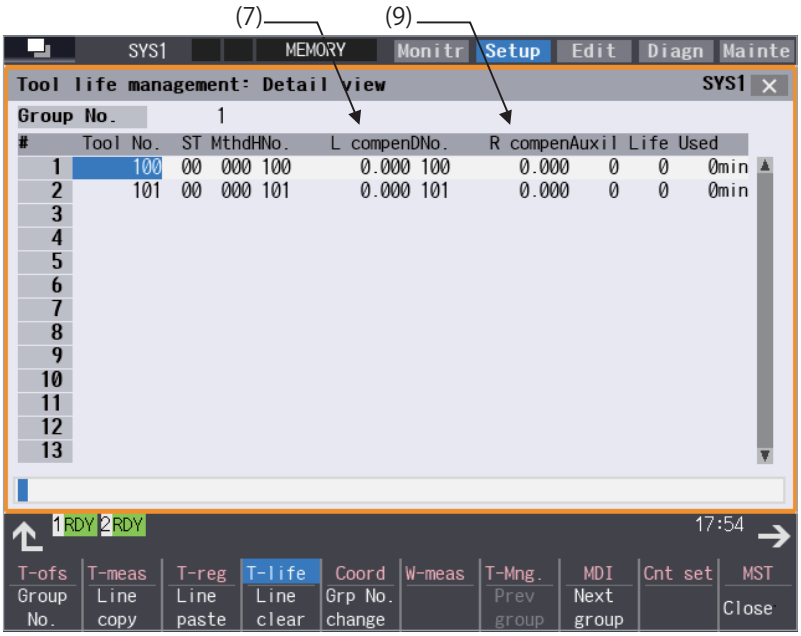
You can register the tool life management data on this screen. The tool usage is displayed on this screen.
Press the menu [T-life group] to change to this screen when the group list is displayed.

Use the PAGE or PAGE key to scroll the screen if all the registered tools cannot be displayed in the display area of the screen.
This screen is common to the tool life managements I, II and III.

<Normal display>



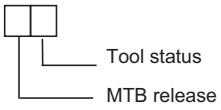
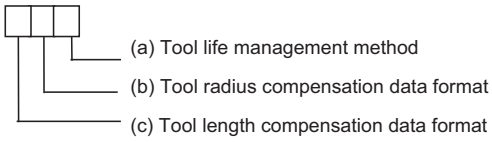
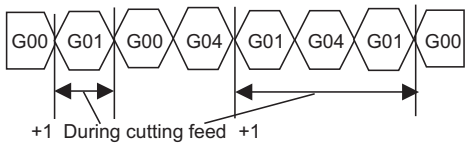
<Detailed display>



Note

(1) When the part system is changed while the data is being edited on the group unit display screen, the editing data will be discarded. To display the group unit of the changed part system, return to the group list display screen first and then display the group unit display screen again.

Display items

Display items	Details	Setting range
(1) Group No.	This is the group No. of the tool which is subject to the tool life management. (*1) 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	 Tool status 0: Unused tool Normally set to "0" when the tool is replaced with a new tool. 1: Tool in use 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: Abnormal tool 1 (*2) 4: Abnormal tool 2 (*2)	
(5) Mthd	 (a) Tool life management method 0: Cumulative cutting hours This is the method to manage by the actual cutting time. 1: Cumulative number of mounting times This is the method to manage by the number of times the tool is treated as a spindle tool at the tool change etc. However, when no cutting feed command (G01, G02, G03 etc.) is issued for the spindle tool, the number of mounting times is not counted. 2: Cumulative number of cuttings This is the method to manage by counting the number of times when the rapid traverse command (G00 etc.) is changed to the cutting feed command (G01, G02, G03 etc.). However, a rapid traverse or a cutting feed command with no movement is not counted. If any command other than the rapid traverse command is issued between two cutting feed commands, the usage data is not counted.  By setting the parameter "#8165 Tlife_DrillCnt" to "1", the usage data is counted by number of drilled holes during the modal of the fixed cycle for drilling.	

Display items	Details	Setting range
(5) Mthd (continued)	(b) Tool radius compensation data format (c) Tool length compensation data format 0: Compensation number format 1: Incremental compensation format 2: Direct compensation format Displays the length dimension or radius dimension of the tool compensation amount of the compensation number designated by "HNo." or "DNo." in the display item "L compen" or "R compen". If the value of the display item "L compen" or "R compen" is changed, the length dimension or radius dimension of the tool compensation amount is rewritten.	
(6) HNo.	Length compensation number. Entering a new tool number automatically assigns the same compensation number as the tool number. If a new tool number is entered that is outside the allowable compensation number range, the compensation number is assigned to "0". The compensation number can also be changed to any number as needed.	1 to 999
(7) L compen	Length dimension of tool compensation amount of the compensation number designated by "HNo.".	(*3)
(8) DNo.	Tool radius compensation number. Entering a new tool number automatically assigns the same compensation number as the tool number. If a new tool number is entered that is outside the allowable compensation number range, the compensation number is assigned to "0". The compensation number can also be changed to any number as needed.	1 to 999
(9) R compen	Radius dimension of tool compensation amount of the compensation number designated by "DNo.".	(*3)
(10) Aux.	This depends on the MTB specifications.	0 to 65535
(11) Life	Set the tool life for each tool to this column based on the format set in the "Mthd" column, selecting from the cutting time (minutes), the number of mounting times (to spindle), or the number of cuttings (holes to be drilled). The tool life is infinite when "0" is set.	Cutting time: 0 to 4000 (min.) Number of mounting times: 0 to 65000 (set) Number of cuttings: 0 to 65000 (set)
(12) Used	This displays the usage data for each tool based on the method specified for the tool life management method.	

(*1) The group No. is not displayed for the tool life management III.

(*2) This depends on the MTB specifications.

(*3) The "L compen" and "R compen" calculation/direct compensation amount setting/display range are as follows based on a combination of settings of "#1003 iunit" (input unit) and "#1041 Initial inch".

#1003 iunit	#1041 Initial inch	Setting range
B	0	-9999.999 to 9999.999
	1	-999.9999 to 999.9999
C	0	-9999.9999 to 9999.9999
	1	-999.99999 to 999.99999
D	0	-9999.99999 to 9999.99999
	1	-999.999999 to 999.999999
E	0	-9999.999999 to 9999.999999
	1	-999.9999999 to 999.9999999

When the input value is set to the one without decimal point, the setting unit follows to the parameter "#8119 Comp. unit switch".

The tool compensation amount to be displayed by "L compen" and "R compen" are switched depending on the tool compensation type as follows:

Tool compensation type	Item	Setting range
I	L compen	The compensation amount that is set in the designated compensation number
	R compen	
II	L compen	The compensation amount that is set in the length of the designated compensation number
	R compen	The compensation amount that is set in the radius of the designated compensation number
III	L compen	The compensation amount that is set in the Z axis of tool length data of the designated compensation number
	R compen	The compensation amount that is set in nose R of the designated compensation number

Note

(1) The tool usage data is not counted in the following cases even when the tool life management is valid.

- The value of "Life" is "0".
- The display of "ST" is "2" (Normal life tool).
  Only when the parameter "#1259 set31/bit0" is set to "0". (*1)
- The display of "ST" is "3" (Abnormal tool 1) or "4" (Abnormal tool 2).
- Miscellaneous function lock
- Dry run
- Single block
- Skip
- Graphic check
- When the automatic machine lock signal is being input at least for one axis.
  If the automatic machine lock signal is turned ON at the middle of the program, the tool usage data is counted on the block where the signal is turned ON. However, the data will not be counted on the subsequent blocks.

(*1) When the parameter "#1259 set31/bit0" is set to "1", and the usage data is counted until when it reaches the maximum value indicated below even if the display of "ST" is "2" (Normal life tool).
Cutting time: 4000 minutes
Number of cuttings: 65000 sets

Menus

Menus	Details
Group No.	This displays the data of a group when that group No. of tool life management data is set and the [INPUT] key is pressed. (*1) The group No. can be referred to in the group list.
Line copy	Copies one line of tool life management data where the cursor is located.
Line paste	The copied tool life management data is written to the data in the line where the cursor is located. If the data is changed after the menu [Line copy] is pressed, the data before change (at the time when the menu [Line copy] is pressed) is pasted. The copied data can be pasted repeatedly until a new data is copied.
Line clear	This deletes the designated tool life management data. The range to be deleted can be specified. Specify the range with the first line No. and the last line No. separated by "/". If the [INPUT] key is pressed without specifying a line, the data in the line where the cursor is located is erased.
Grp No. change	This changes a group No. (*1) If a new group No. is set and the [INPUT] key pressed, the group No. changes. An error occurs if an existing group No. is specified.
Disp details	Displays the pop-up window for details of the selected group.
Prev group	Displays the previous group No. data. (*1)
Next group	Displays the next group No. data. (*1)
Group list	This displays the list of tool life data groups. (*1)
Group regist	This creates a new group. (*2)
Group delete	This deletes all tool life management data contained in the currently displayed group No. (*2)
Close	This closes the pop-up window for details of the selected group.

(*1) This menu is not valid for the tool life management III.

(*2) This menu is not displayed for the tool life management III.

Operation method (Designating a group No.)

(1) Press the menu [Group No.].

(2) Enter the group number to be displayed in the input area, and press the [INPUT] key.

Note

(1) 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 [Y] or [INPUT] is pressed.

Operation method (Changing a group No.)

(1) Press the menu [Grp No.change].

(2) Enter the group number to be replaced in the input area, and press the [INPUT] key.

Note

(1) An error occurs if the group No. duplicates a pre-existing group No.

Operation method (Setting the tool life management data)

(1) Move the cursor to the data to be set using the PAGE, PAGE, [↑], [↓], [|←] or [→|] key.
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 the tool life management data in the input area, and press the [INPUT] key.

Note

(1) When no value is set in the status ("ST"), the setting is invalid.
(2) The same tool cannot be registered in more than one group.
(3) If the tool No. is not set, the other data cannot be set.
(4) Change the "Mthd" to initialize the related items.
When the tool life management method (the 3rd digit) is changed, the data in "Life" and "Usage" are cleared to "0", and the setting units are also changed.
When tool radius compensation data format (the 2nd digit) is changed, the data in "R compen" is cleared to 0".
When tool length compensation data format (the 1st digit) is changed, the data in "L compen" is cleared to 0".
The number of digits after the decimal point is also changed depending on the specifications.

Operation method (Copying/pasting the tool life management data)

(1) Move the cursor to the line to be copied.

(2) Press the menu [Line copy].

(3) Move the cursor to the line where the data is to be pasted.

(4) Press the menu [Line paste].

Note

(1) It is not possible to paste into a line for which a tool No. has not been set.
(2) Once copied, the data is held until a new data item is copied.

Operation method (Deleting the tool life management data by one line)

(1) Move the cursor to the line to be deleted.

(2) Press the menu [Line clear] and [INPUT] keys.

(3) Press the [Y] or [INPUT] key.
Press a key other than [Y] or [INPUT] in order not to delete the data.

Operation method (Designating and deleting the multiple lines of data)

(1) Press the menu [Line clear].

(2) Set the range to be deleted by marking the first and last # Nos. with a "/".
(Example) 122/125 [INPUT]

(3) Press the [Y] or [INPUT] key.
Press a key other than [Y] or [INPUT] in order not to delete the data.

Operation method (Registering a group)

(1) Press the menu [Group regist].

(2) Enter the group number to be registered in the input area, and press the [INPUT] key.

Note

(1) An error occurs if the group No. duplicates a pre-existing group No.

(2) The group is not registered if one of the operations below is performed after registering a new group No.

- When you quit this operation without registering even 1 item of the tool life management data.
- When the screen is changed to that of other than the group unit display.
- When the part system is changed.

Operation method (Deleting a group)

(1) Move the cursor to the group No. to be deleted.

(2) Press the menu [Group delete] and [INPUT] key.

(3) Press the [Y] or [INPUT] key.

6.6.1.2 Operation Descriptions (Tool life management I/III)

The tool No. which is to be used is specified by T command with this management type.
A tool which is mounted to the spindle by the tool change command (M06) is subject to the tool life management.
The tool usage is monitored based on the tool life management method (cumulative cutting hours, cumulative number of mounting times or cumulative number of cuttings) which is set in the "Mthd" column on the group unit display screen.

When the use of the tool is started, "1" which indicates the status of "Tool in use" is displayed in the "ST" column. The cumulative cutting hours (or the cumulative number of cuttings) is displayed in the "Used" column.
When the cumulative cutting hours (or the cumulative number of cuttings) has reached the value set in the "Life" column, the display of the "ST" column changes to "2" which indicates the status of "Normal life tool".

[Display example of the "Mthd" column on the group unit display screen]

When "0" (management by the cumulative cutting hours) is set for the tool life management method, and the tool of No. 2 has been used for one minute.

Group No. 1

#	Tool No.	ST	Mthd	HNo.	DNo.	Auxil	Life	Used
1	1	00	000	1	1	0	120	0min
2	2	01	000	2	2	0	120	1min
3								

Tool compensation operation

The tool is compensated by the amount of the tool compensation No. specified by either of the following addresses in the machining program:
- Address H of the G43/G44 command (tool length compensation)
- Address D of the G41/G42 command (tool radius compensation)

The first 2 digits of "Mthd" (tool radius compensation data format/tool length compensation data format), "HNo." and "DNo." tool life management screen are not used for the tool compensation.

6.6.1.3 Operation Descriptions (Tool life management II)

Specify the tool to be used by issuing the T command.
Either of the following command methods can be selected for T command of the tool life management II:
- The method to designate the group No. in which the target tool is registered
- The method to designate the tool No. of the target tool

It depends on the MTB specifications which method is to be selected. (The setting can be changed by the combination of the parameters "#1104 T_Com2" and "#1227 aux11/bit1".)

- ♦When the group No. is designated
  This function selects a tool in accordance with the setting of the parameter "#1105 T_sel2" from the group which is designated by the T command, and the selected tool is set to the spare tool.
- ♦When the tool No. is designated
  This function searches the group in which the tool, whose No. is designated by the T command, is registered. And then a tool is selected in accordance with the setting of the parameter "#1105 T_sel2" from the group, and the selected tool is set to the spare tool.
  After the selected tool is mounted to the spindle by the tool change command (M06), it will be managed by the tool life management.

The tool usage is monitored based on the tool life management method (cumulative cutting hours, cumulative number of mounting times or cumulative number of cuttings) which set in the "Mthd" column on the group unit display screen.
When the use of the tool is started, "1" which indicates the status of "Tool in use" is displayed in the "ST" column. The cumulative cutting hours (or the cumulative number of cuttings) is displayed in the "Used" column.
When the cumulative cutting hours (or the cumulative number of cuttings) has reached the value set in the "Life" column, the display of the "ST" column changes to "2" which indicates the status of "Normal life tool".

[Display example of the group unit display screen]

When the tool of No.11 which is selected from the group 2 has been used for one minute.

Group No. 2

#	Tool No.	ST	Mthd	HNo.	DNo.	Auxil	Life	Used
1	11	01	000	11	11	0	120	1min
2	12	00	000	12	12	0	120	0min
3								

When the tool life management method is set "0" (management by the cumulative cutting hours) in the "Mthd" column, the display of the "Life" and the "Used" columns are based on time.

Tool compensation operation

■When the parameter "#1227 aux1/bit1" = 0

The tool compensation is performed based on the setting value of the "Mthd" on the tool management screen.
The tool compensation data is acquired from the tool compensation screen regardless of the setting of the "Mthd".

0: Compensation number format

The G43/G44 command executes the compensation using the data (the tool length and the length wear) corresponding to "HNo." of the selected tool.
The G41/G42 command executes the compensation using the data (the radius dimension and the radius wear) corresponding to "DNo." of the selected tool.

1: Incremental compensation format

The G43/G44 command executes the compensation using the data (the tool length and the length wear) of the compensation No. specified by the address H, by adding the data (the tool length) corresponding to "HNo." of the selected tool to the said data.
The G41/G42 command executes the compensation using the data (the radius dimension and the radius wear) of the compensation No. specified by the address D, by adding the data (the radius dimension) corresponding to "DNo." of the selected tool to the said data.

2: Direct compensation format

The G43/G44 command executes the compensation using the data (the tool length) corresponding to "HNo." of the selected tool.
The G41/G42 command executes the compensation using the data (the radius dimension) corresponding to "DNo." of the selected tool.

Note

(1) The value specified by the address H of the G43/G44 command or the address D of the G41/G42 command is not used in the compensation number format or the direct compensation format.

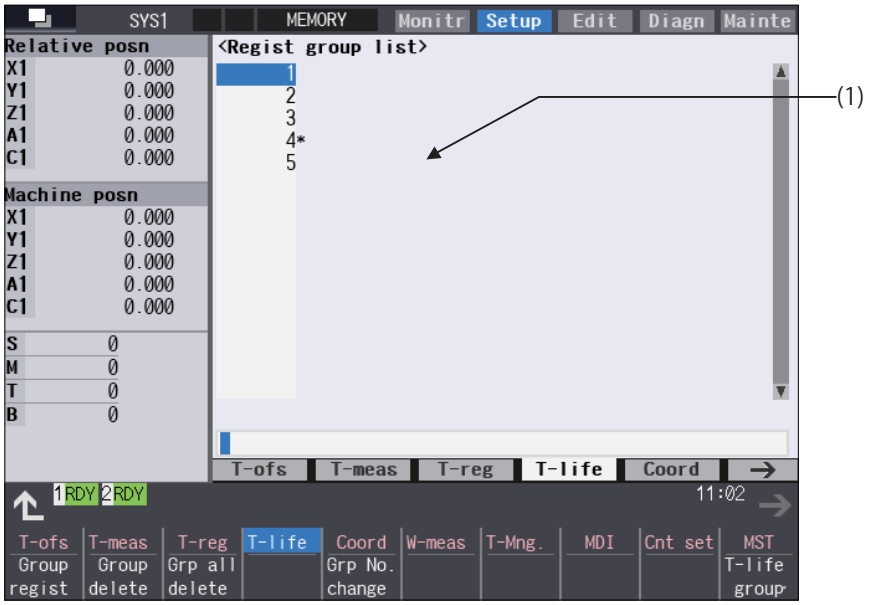
■When the parameter "#1227 aux1/bit1" = 1

The tool compensation operation varies as follows depending on the value of the address H or the address D.

Condition	Tool compensation operation
Address H of the G43/G44 command is "H99".	The G43/G44 command executes the compensation using the data (the tool length and the length wear) corresponding to "HNo." of the selected tool.
Address H of the G43/G44 command is other than "H99".	The G43/G44 command executes the compensation using the data (the tool length and the length wear) of the compensation No. specified by the address H.
Address D of the G41/G42 command is "H99".	The G41/G42 command executes the compensation using the data (the radius dimension and the radius wear) corresponding to "DNo." of the selected tool.
Address D of the G41/G42 command is other than "D99".	The G41/G42 command executes the compensation using the data (the radius dimension and the radius wear) of the compensation No. specified by the address D.

6.6.1.4 Registering the Tool Life Management Data Groups on the Group List Display Screen (Tool life management I/II)

Tool life management data groups can be registered and deleted.
Press the menu [Group list] to change to this screen when the group unit is displayed.



Display items

Display items		Details
(1)	Regist group list	The group Nos. registered as the tool life management data are listed in ascending order. Group Nos. can be set from 1 to 99999999, and maximum 1000 groups can be registered. Maximum 60 groups can be displayed at a time. Use the PAGE or PAGE key to scroll the screen if the No. of registered groups exceeds 60. 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 specifications. •In tool life management II, when all the tools' statuses (ST) of the target group become 2 to 4, "*" is displayed.

Menus

Menus	Details
Group regist	Creates a new group and adds the group No. to the list.
Group delete	This deletes 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 deleted.
Grp all delete	This deletes all groups and their tool life management data registered in the part system which is currently selected.
Grp No. change	This changes a group No. Specify a new group No. and press the [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.
T-life group	The tool life management data of the group No. where the cursor is located is displayed. The tool life management data can be set and displayed on a group basis on this mode.

Operation method (Registering a group)

(1) Press the menu [Group regist].

(2) Set the No. of the group to be registered.
(Example) 25 [INPUT]

Note

(1) An error occurs if the group No. duplicates a pre-existing group No.

(2) The group is not registered if one of the operations below is performed after registering a new group No.

- When you quit this operation without registering even 1 item of the tool life management data.
- The screen is changed to that of other than the group unit display.
- The part system is changed.

Operation method (Deleting a group)

(1) Press the menu [Group delete].

(2) Designate the group to be deleted. → The selected group No. is deleted.
(Example) 72 [INPUT]

If nothing is input in the input area, the group at the cursor location is deleted.

Operation method (Deleting all groups)

(1) Press the menu [Grp all delete].

(2) Press the [Y] or [INPUT] key. → All the registered groups and their tool life management data is deleted.

Note

(1) If no group has been registered, the operation message "Can't delete all groups" appears.

Operation method (Changing a group No.)

(Example) Changing the group No. from 5 to 20

(1) Use the [↑], [↓], PAGE or PAGE key to move the cursor to the group No. 5.

(2) Press the menu [Grp No.change]. → The menu is highlighted.

(3) Designate the group No. to be replaced. → The group No. changes to "20".
20 [INPUT]

Operation method (Changing to the group unit display)

(Example) Displaying the tool life management data of the group No. 5.

(1) Use the [↑], [↓], PAGE or PAGE key to move the cursor to the group No. 5.

(2) Press the menu [T-life group] or [INPUT] keys. → The screen changes to the group unit display and the group No. "5" tool life management data displays.

6.6.2 Tool Life Management (L System)

Select a type of the tool life management and register tool life data, and then issue the T command (tool command) to start the tool life management.

■Selecting the tool life management type

The tool life management I: The usage of a tool commanded by the machining program is monitored.

The tool life management II: Command a group No. in which the same type of the tool is registered by the machining program, and the corresponding tool is selected form the group to monitor its usage.

■Registering the tool life data

Register the tool life data which is to be used on the tool life management screen. The tool life data is set by the estimated tool life (hours) or by the estimated number of cuttings.

■Executing the tool life management

The tool life management is executed during the operation by the machining program.

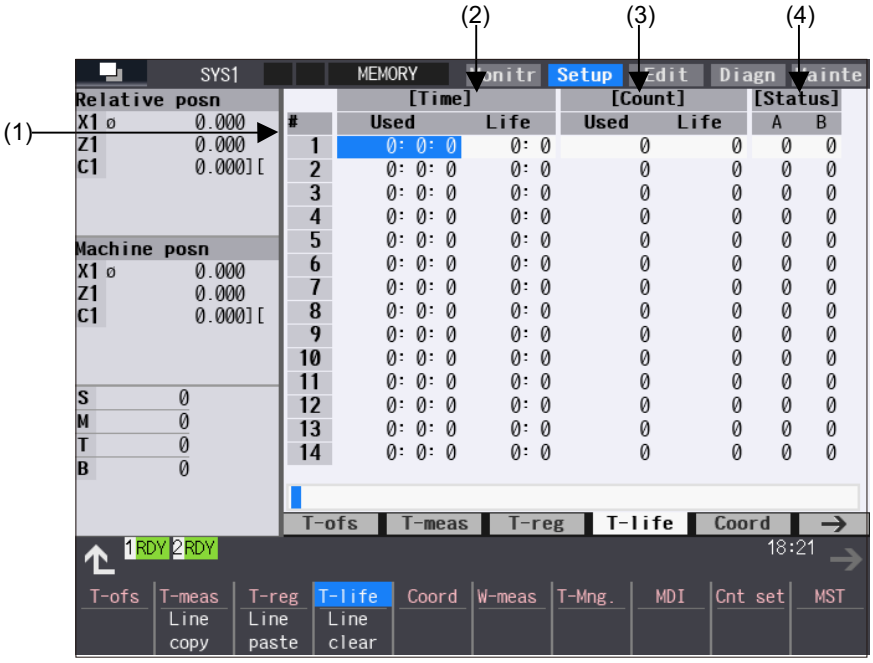
T command (tool command) is issued based on the tool life management type of the tool life.

Management type	Command method	Details
Tool life management I	Specifies the tool No.	T ○○ΔΔ command (○○: No., ΔΔ: compensation No.)
Tool life management II	Specifies the group No.	T □□□□99 command (□□□□: group No.)

6.6.2.1 Registering the Tool Life Management Data (Tool Life Management I)

The tool life management data can be set and displayed.

Use the PAGE or PAGE key to scroll the screen if all the registered tools cannot be displayed in the display area of the screen.



Display items

Display items		Details	Setting range
(1) #		Tool No. The # No. is highlighted in the following cases. - The cumulative cutting hours have reached the estimated tool life (hours). - The cumulative number of cuttings exceeds the estimated number of cuttings.	
(2) Time	Used	The cumulative cutting hours of the tool The count goes up during cutting.	0:0 to 99:59 (hour:minute) (*1) 0:0 to 5000:00 (hour:minute) (*2)
	Life	Setting for the estimated tool life Set the estimated maximum tool life.	0:0 to 99:59 (hour:minute) (*1) 0:0 to 5000:00 (hour:minute) (*2) (0:0 = no warning given)
(3) Count	Used	The cumulative number of cuttings for tool use The count goes up during cutting. By setting the parameter "#8165 Tlife_DrillCnt" to "1", the usage data is counted by number of drilled holes during the modal of the fixed cycle for drilling.	0 to 65000 (times) (*1) 0 to 99999999 (times) (*2)
	Life	Setting for estimated number of cuttings Set the estimated maximum number of cuttings.	0 to 65000 (times) (*1) 0 to 99999999 (times) (*2) (0:0 = no warning given)
(4) Status	A	The tool life management status is indicated. 0: Not used 1: Current tool (tool being used) 2: Estimated tool life hours are exceeded.	0 to 2
	B	(Used by MTB)	0 to 99

(*1) When the parameter "#1284 ext20/bit4" is set to "0"

(*2) When the parameter "#1284 ext20/bit4" is set to "1"

Menus

Menus	Details
Line copy	Copies one line of tool life management data where the cursor is located.
Line paste	The copied tool life management data is written to the data in the line where the cursor is located. If the data is changed after the menu [Line copy] is pressed, the data before change (at the time when the menu [Line copy] is pressed) is pasted. The copied data can be pasted as many times as is required until new data is copied.
Line clear	This deletes the designated tool life management data. The range to be deleted can be specified. Specify the range with the first line No. and the last line No. separated by "/". If the [INPUT] key is pressed without specifying a line, the data in the line where the cursor is located is erased.

Operation method (Setting the tool life management data)

(1) Move the cursor to the data to be set using the PAGE, PAGE, [↑], [↓], [|←] or [→|] key.
It is possible to move to the line after the registered line.

(2) Enter the value in the input area, and then press the [INPUT] key.

Operation method (Copying/pasting the tool life management data)

(1) Move the cursor to the line to be copied.

(2) Press the menu [Line copy].

(3) Move the cursor to the line where the data is to be pasted.

(4) Press the menu [Line paste].

Note

(1) Once copied, the data is held until a new data item is copied.

Operation method (Deleting the tool life management data by one line)

(1) Move the cursor to the line to be deleted.

(2) Press the menu [Line clear] and [INPUT] keys.

(3) Press the [Y] or [INPUT] key.
Press a key other than [Y] or [INPUT] in order not to delete the data.

Operation method (Designating and deleting the multiple lines)

(1) Press the menu [Line clear].

(2) Set the erasing range by marking the first and last # Nos. with a "/", and press the [INPUT] key.
(Example) 122/125 [INPUT]

(3) Press the [Y] or [INPUT] key.
Press a key other than [Y] or [INPUT] in order not to delete the data.

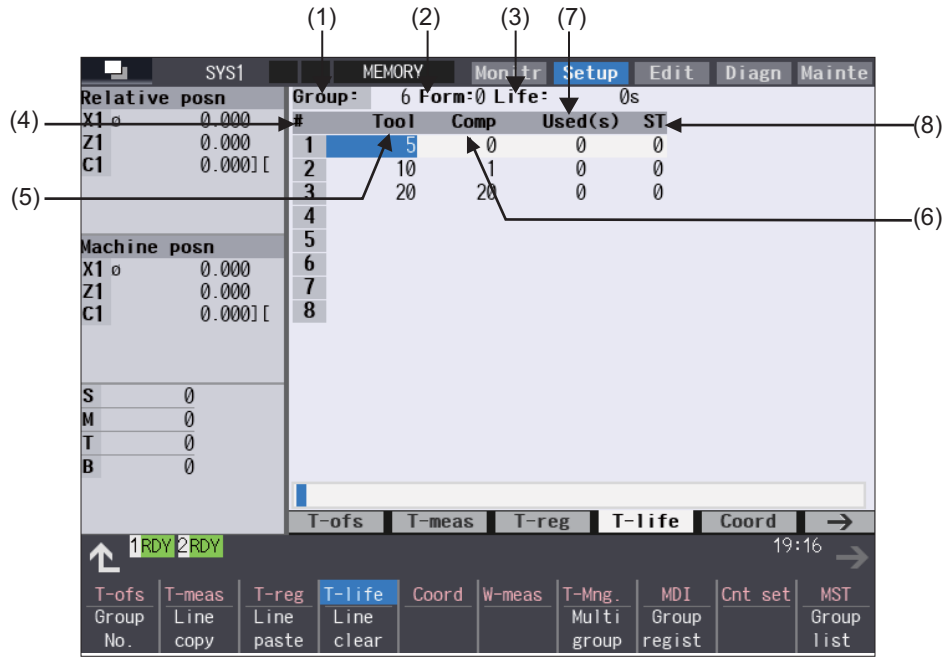
Note

(1) If the [INPUT] key is pressed without setting a # No., the line at the cursor is deleted.

6.6.2.2 Registering Tool Life Management Data on the Group Unit Display Screen (Tool Life Management II)

You can register the tool life management data on this screen. The tool usage is displayed on this screen.
Press the menu [T-life group] to change to this screen when the group list is displayed.

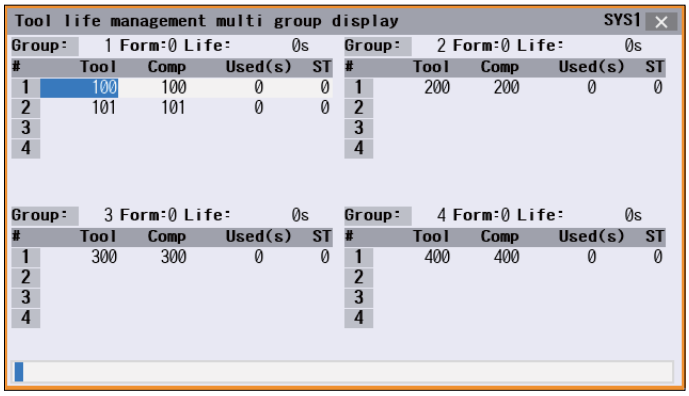
Use the PAGE or PAGE key to scroll the screen if all the registered tools cannot be displayed in the display area of the screen.



The number of groups to be displayed on a single screen depends on the setting of the parameter "#1107 Tlifsc". (This depends on the MTB specifications.)

Setting value of #1107	No. of display groups	Max. No. of registered tools
0	1	16
1	2	8
2	4	4

[Display example: 4 groups are displayed on a single screen (#1107 Tllfsc = 2)]



Display items

<Group information>

Display items	Details	Setting range
(1) Group	Life management group No.	1 to 9999
(2) Method	This displays whether the group currently being displayed is managed by the cumulative cutting hours or the cumulative number of cuttings.	0: Cumulative cutting hours 1: Cumulative number of cuttings
(3) Life	The tool life management data for the currently displayed group is set in this column. The following data and units are displayed based on the management method. Estimated tool life: (s) Estimated number of cuttings: (set)	Estimated tool life: 0 to 99999999 (sec) Estimated number of cuttings: 0 to 999999 (set)

<Tool Life management data in a group>

Display items	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.	The tool No. is set in this column.	1 to 999999
(6) Comp No.	The compensation No. is set in this column.	1 to 80
(7) Used	Estimated too life or estimated number of cuttings is set in this column. When the setting is omitted, "0" is displayed. The following units for the usage data is displayed based on the management method. Cutting time: (s) Number of cuttings: (set)	Estimated tool life: 0 to 99999999 (sec) Estimated number of cuttings: 0 to 999999 (set)
(8) ST	This sets the tool status. When the setting is omitted, "0" is displayed. 0: Unused tool 1: Used tool 2: Normal life tool 3: Skipped tool	0 to 3

Menus

Menus	Details
Group No.	This displays the data of a group when that group No. of tool life management data is set and the [INPUT] key is pressed. The group No. can be referred to in the group list.
Line copy	Copies one line of tool life management data where the cursor is located.
Line paste	The copied tool life management data is written to the data in the line where the cursor is located. If the data is changed after the menu [Line copy] is pressed, the data before change (at the time when the menu [Line copy] is pressed) is pasted. The copied data can be pasted as many times as is required until new data is copied.
Line clear	This deletes the designated tool life management data. The range to be deleted can be specified. Specify the range with the first line No. and the last line No. separated by "/". If the [INPUT] key is pressed without specifying a line, the data in the line where the cursor is located is erased.
Grp No. change	This changes a group No. If a new group No. is specified and the [INPUT] key pressed, the group No. changes. An error occurs if an existing group No. is specified.
Group change	The group active area is changed if parameter "#1107 Tllfsc" is set to "1" or "2".
Multi group	Changes to the multiple group life management data display mode based on the parameter "#1107 Tllfsc" setting value.
Group regist	This creates a new group, and the group is displayed with the tool life management data empty.
Group list	This displays the group list of tool life management data.
Close	The multiple group display mode of the tool life management data is canceled.

Operation method (Designating a group No.)

(1) Press the menu [Group No.].

(2) Enter the group number to be displayed in the input area, and press the [INPUT] key.

Note

(1) 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 [Y] or [INPUT] 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 (cutting time), and tool life data = 0.

Operation method (Changing a group No.)

(1) Press the menu [Grp No.change].

(2) Enter the group number to be replaced in the input area, and press the [INPUT] key.

Note

(1) An error occurs if the group No. duplicates a pre-existing group No.

Operation method (Setting the tool life management data)

(1) Move the cursor to the data to be set using the PAGE, PAGE, [↑], [↓], [|←] or [→|] key.
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 the tool life management data in the input area, and press the [INPUT] key.

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 management data)

(1) Move the cursor to the line to be copied.

(2) Press the menu [Line copy].

(3) Move the cursor to the line where the data is to be pasted.

(4) Press the menu [Line paste].

Note

(1) Once copied, the data is held until a new data item is copied.

Operation method (Deleting the tool life management data by one line)

(1) Move the cursor to the line to be deleted.

(2) Press the menu [Line clear] and [INPUT] keys.

(3) Press the [Y] or [INPUT] key.
Press a key other than [Y] or [INPUT] in order not to delete the data.

Operation method (Designating and deleting the multiple lines)

(1) Press the menu [Line clear].

(2) Set the range to be deleted by marking the first and last # Nos. with a "/".
(Example) 1/5 [INPUT]

(3) Press the [Y] or [INPUT] key.
Press a key other than [Y] or [INPUT] in order not to delete the data.

Note

(1) If the [INPUT] key is pressed without setting a # No., the line at the cursor is deleted.

Operation method (Registering a group)

(1) Press the menu [Group regist].

(2) Set a group No., the management method and the tool life to register separated with "/".
The tool life management method is to be set in the "Form" column.
Set it to "0" to manage by cutting hours, and set it to "1" to manage by number of cuttings.
Set the tool life data as follows:
When "0" is set to the "Form" column, set the estimated tool life (hours).
When "1" is set to the "Form" column, set the estimated number of cuttings.
(Example) 25/1/3000 [INPUT]

Note

(1) If the method and life data are omitted, a new group is created with method = 0 (cutting time), and tool life data = 0.
(2) An error occurs if the group No. duplicates a pre-existing group No.
(3) The group is not registered if one of the operations below is performed after registering a new group No.
- When you quit this operation without registering even 1 item of the tool life management data.
- 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 display)

(1) Press the menu [Multi group].

(2) Press the menu [Group change]. → The cursor indicating the active status moves to the next group.

6.6.2.3 Operation Descriptions (Tool life management I)

The tool No. which is to be used is specified by T command with this management type. The commanded tool is subject to the tool life management.
When the machining using the tool is started, "1" which indicates the status of "Tool in use" is displayed in the "Status" column. The cumulative cutting hours (or the cumulative number of cuttings) is displayed in the "Used" column.
When the cumulative cutting hours or the cumulative number of cuttings has reached the value set in the "Life" column, the display of the "A" column changes to "2" which indicates "tool life end". (*1)

(*1) The status changes to "2" as soon as the cumulative cutting hours reaches the estimated tool life.
The status changes to "2" by the T command after the cumulative number of cuttings has reached the estimated number of cuttings.

6.6.2.4 Operation Descriptions (Tool life management II)

The tools to be used are classified into several groups and their tool lives are monitored on a group basis (by the cumulative cutting hours or the cumulative number of cuttings).
When the tool which is currently being used has reached the end of its tool life, a spare tool in the same group is selected to exchange automatically at the timing of the next T command. This function enables unmanned operation over a long time.

■T command (tool command)

Follow the format below to specify the group No. in the T command.

T□□□□99: Start to use the tool in the group □□□□.
T□□□□88: Cancel the offset of the tool in the group □□□□.
(This is the same as "T○○00"; ○○ indicates the tool currently being used.)

When the tool No. is specified by "T○○ΔΔ", the tool life management is not performed. (○○: tool No., ΔΔ: compensation No.)

■Tool selection

When multiple tools are registered in the same group, a spare tool is selected in the order of the registration.
The tool change is performed at the timing when the "T□□□□99" is issued after the tool currently being used has reached the end of its life.
When the tool currently being used has not reached the end of its tool life, the tool change is not performed.
When the tool change cannot be performed because all the tools in the group have reached the end of their tool lives, the tool life management function continues counting for the group.

■Tool compensation No. selection

Multiple tool compensation Nos. can be registered to a single tool in the same group with the tool life management II.
Command the "T□□□□99" again to change the compensation No.
Command the M code for re-counting to return to the first compensation No.

[Example when multiple compensation Nos. are registered to a single tool]

Group: 1 Form:0 Life: 300s

#	Tool	Comp	Used(s)	ST
1	17	1	0	0
2	17	2	0	0
3	17	3	0	0
4	21	4	0	0
5				

Machining program		Tool selection
T0199;	…	Selects the compensation No. 1. of the tool of No. 17.
:		
T0199;	…	Selects the compensation No. 2. of the tool of No. 17.
:		
T0199;	…	Selects the compensation No. 3. of the tool of No. 17.
:		
T0199;	…	Selects the compensation No. 3. of the tool of No. 17.
:		Hereafter, selects the compensation No. 3. of the tool of No. 17 until this tool has reached the end of its tool life.
(The tool of No. 17 has reached the end of its tool life.)		
:		
T0199;	…	Selects the compensation No. 4. of the tool of No. 21.
:		

■Estimated tool life/maximum estimated number of cycles

The estimated tool life and the maximum estimated number of cycles in tool life management II are as follows.
Estimated tool life: 99999999 seconds (approx. 27777 hours)
Maximum estimated number of cycles: 999999 times

Note

(1) When you register or change the tool life management data in the selected tool group, command "T□□□□99" again to re-select the group. If a new registration or a change of the data is performed in the state that any tool is being selected and then the machining operation continues, the tool life management may not function properly.

(2) "T□□□□99" or "T□□□□88" cannot be commanded when the parameter "#1441 Tcode_Method_Chg" is set to "1". When "T□□□□99" or "T□□□□88" is commanded, it is regarded as "T○○ΔΔ" command. (○○: tool No., ΔΔ: compensation No.)

Counting the cumulative cutting hours/the cumulative number of cuttings

There are two methods for the tool life management II; the management by the cumulative cutting hours or the management by the cumulative number of cuttings.
For the management method by the cumulative number of cuttings, it depends on the MTB specifications how and when the cumulative number will be incremented. (They are determined by the setting of the parameter "#1277 ext13/bit0".)
The cumulative number is not incremented when the display of the "ST" column on the tool management screen is "3" (Abnormal tool 1) or "4" (Abnormal tool 2).

Re-count M code

When multiple compensation Nos. are registered for a single tool, the first compensation No. can be selected with the re-count M code.
The re-count M code can be set by the parameter "#1108 TlrectM". The re-count M code is invalid when the setting value is "0". Do not set "1", "2" or "30".

Note

(1) For the management method by the cumulative number of cuttings, the cumulative number will be incremented once when the machining is performed after the re-count M code is commanded.

(2) For the management method by the cumulative cutting hours, the cumulative number is not changed even though the re-count M code is commanded, because the machining time is cumulated with this method.

Status [ST]

For the tool life management II, the data of the cumulative cutting hours (or the cumulative number of cuttings) is saved for each registered tool.
However, when several tools of the same No. are registered as shown the figures below, the tool life is managed by the total amount of the cumulative cutting hours (or the cumulative number of cuttings) of each tool. Thus, the display of the "ST" column for the tool, which has not reached the end of its tool life yet, may be "2" ("Normal life tool").

Management by cutting hours (Life: 300 seconds)

Group: 1 Form:0 Life: 300s

#	Tool	Comp	Used(s)	ST
1	17	1	150	2
2	17	2	151	2
3	17	3	0	2
4	21	4	0	0
5				

Management by number of cuttings (Used: 20 times)

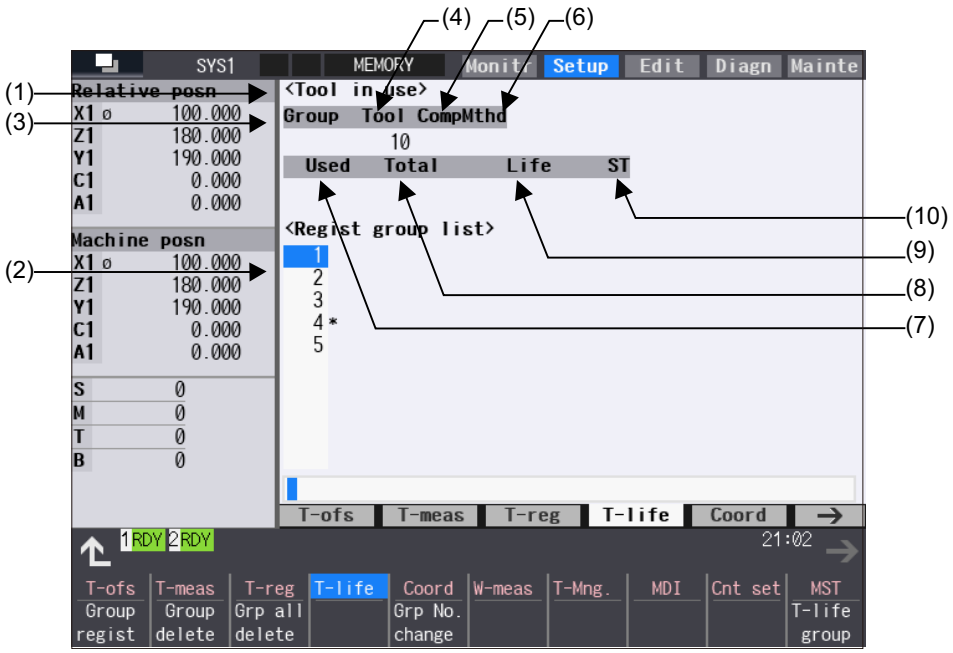
Group: 1 Form:1 Life: 20set

#	Tool	Comp	Used(set)	ST
1	17	1	10	2
2	17	2	11	2
3	17	3	0	2
4	21	4	0	0
5				

The display of the "ST" column on the tool life management screen is updated to "2" when the NC is reset after the cumulative number of cuttings has reached the estimated number.

6.6.2.5 Registering the Tool Life Management Data Groups on the Group List Display Screen (Tool Life Management II)

Tool life management data groups can be registered and deleted. The tool life management method and the tool life data can also be set in this screen.
Press the menu [Group list] to change to this screen when the group unit is displayed.



Display items

Display items		Details
(1)	Tool in use	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 specifications.

<Tool in use>

Display items		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)	Method	Displays whether the group currently being used is controlled by the cutting time or number of cuttings.	0: Cumulative cutting hours 1: Cumulative number of cuttings
(7)	Used	Displays the usage data for the tool currently being used.	Cumulative cutting hours: 0 to 99999999 (sec) Cumulative number of cuttings: 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.	
(9)	Life	Displays the life value for the tool currently being used.	
(10)	ST	Displays the status of the tool currently being used.	0: Unused tool 1: Used tool 2: Normal life tool 3: Skipped tool

Menu items and operation methods are the same as for M system. Refer to "6.6.1.4 Registering the Tool Life Management Data Groups on the Group List Display Screen (Tool life management I/II)".

6.6.3 Tool Life Prediction Function

In tool life management II for L system, by setting the remaining tool life prediction setting value (tool life time or the number of cuttings) in advance, when the remaining tool life of the tool in use "tool life data - tool usage data" (tool life time or the number of cuttings) is below the set value, the "Tool life prediction" signal (XC96) can be output.

To use this function, set the parameter "#1277 ext13/bit1" to "1". The output conditions of the tool life prediction signal can be changed with the parameters "#1277 ext13/bit2" and "#1277 ext13/bit3".

#1277 ext13		Output conditions of "Tool life prediction" signal (XC96) and behavior
bit2	bit3	
0	0	The signal is output for each tool when the remaining tool life matches the remaining tool life prediction setting value.
1	0	The signal is output for each tool when the remaining tool life is equal to or less than the remaining tool life prediction setting value.
0	1	The signal is output when the remaining tool life of the last tool in the group matches the remaining tool life prediction setting value.
1	1	The signal is output when the remaining tool life of the last tool in the group is equal to or less than the remaining tool life prediction setting value.

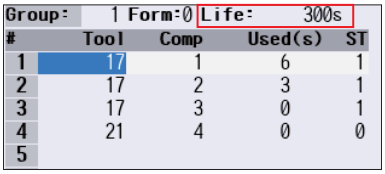
Example of operation

The following describes the example of outputting the tool life prediction signal when the remaining tool life is equal to or less than the remaining tool life prediction setting value.
(When the parameter "#1277 ext13/bit2" is set to "1" and "#1277 ext13/bit3" is set to "0")

(1) Tool life data registration

Command the remaining tool life prediction (address R) when registering the tool life management data by the machining program (G10 L3 command).
Refer to "Programming Manual" for details on the G10 L3 command.

[Tool life]
Group 1: Tool life time 300 s (5 min)
[Remaining tool life prediction]
Group 1: Remaining tool life prediction time 120 s (2 min)
The remaining tool life prediction setting value is not displayed on the screen.



[Machining program]

```
G10 L3;
P1 L5 N0 R2;
T1701;
T1702;
T1703;
T2104;
G11;
M30;
```

… Tool life: 5 min
Management method: time
Remaining tool life prediction: 2 min

(2) Tool life count start

Specify the group number with T□□99 to start using the tool.
The cutting time is counted according to the time of using the tool in the subsequent cutting feeds (G01, G02, G03, G31, G33, etc.).

[Machining program]

```
:
T0199;   … Start to use the tool in the group 01
G01;     … Start to count the cutting time
:
```

(3) Tool life prediction signal output

When the remaining tool life time "tool life time - cutting time" of the tool 17 registered in the group 1 is equal to or less than the remaining tool life prediction time (120 s), the "Tool life prediction" signal (XC96) is output.

6.6.4 Tool Life Management Set Allocation to Part Systems

The number of tool life management sets can be set per part system.
This function is divided into following methods and which one is used depends on the MTB specifications (parameters "#1439 Tlife-SysAssign", "#12055 Tol-lifenum").

Arbitrary allocation: Arbitrarily allocates the number of tool life management sets to each part system.

Fixed allocation: Automatically and evenly allocates the number of tool life management sets to each part system.

The arbitrary allocation enables the efficient allocation because when a certain part system needs only a small number of tool life management sets, the rest can be allocated to another part system.

Fixed allocation

Set the parameter "#1439 Tlife-SysAssign" (Part system allocation of life management tools) to "0" to evenly allocate the number of the tool life management sets to each part system.

Arbitrary allocation

Set the parameter "#1439 Tlife-SysAssign" (Part system allocation of life management tools) to "1" to arbitrarily allocate the number of the tool life management sets to each part system.
The number to be allocated for each part system can be set in the parameter "#12055 Tol-lifenum".

Precautions

(1) For 1-part system, up to the number of tool life management sets in the system is available regardless of the parameter setting.

(2) Even when the specification allows arbitrary allocation, fixed allocation is applied if the parameter "#12055 Tollifenum" = 0 for all part systems.

(3) When the specification allows arbitrary allocation, if the total number of the tool life management sets which set in the parameter "#12055 Tol-lifenum" exceeds that which set in the system, the alarm "Y05 Initial parameter error" occurs.

(4) When the tool life management data, which exceeds the number of the tool life management sets, is input to the file, the message "Can't write file for dev B" at the time when the input data to be input exceeds that of current tool life management sets. The remaining data cannot be input. The unregistered data needs to be set again.

(5) When the number of the tool life management sets is changed, erase all the tools (*1) on the maintenance screen, then reboot the NC and register the tool life data again.

(*1) When all the tools are erased, the tool compensation data is also erased. Output the tool offset data file (TOOL.OFS) before erasing all the tools to back up the tool compensation data.

(6) For fixed allocation, when the number of the tool life management sets is 200 and the parameter "#1046 T-ofs disp type" (Tool compensation type switch) (M system) is set to "1", the number of tool life management sets in the system is 99.

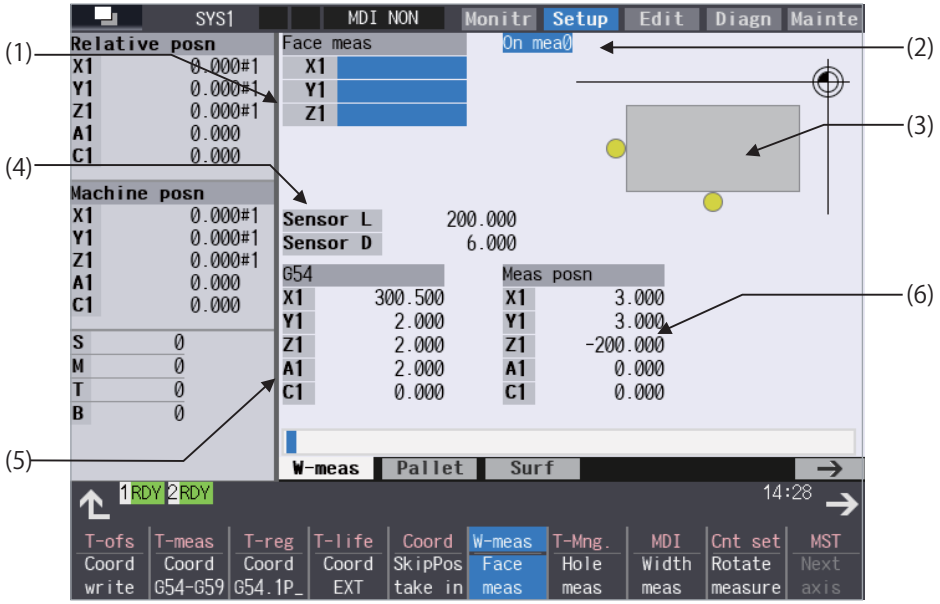
6.7 Setting and Measuring a Workpiece

6.7.1 Workpiece Measurement (M System)

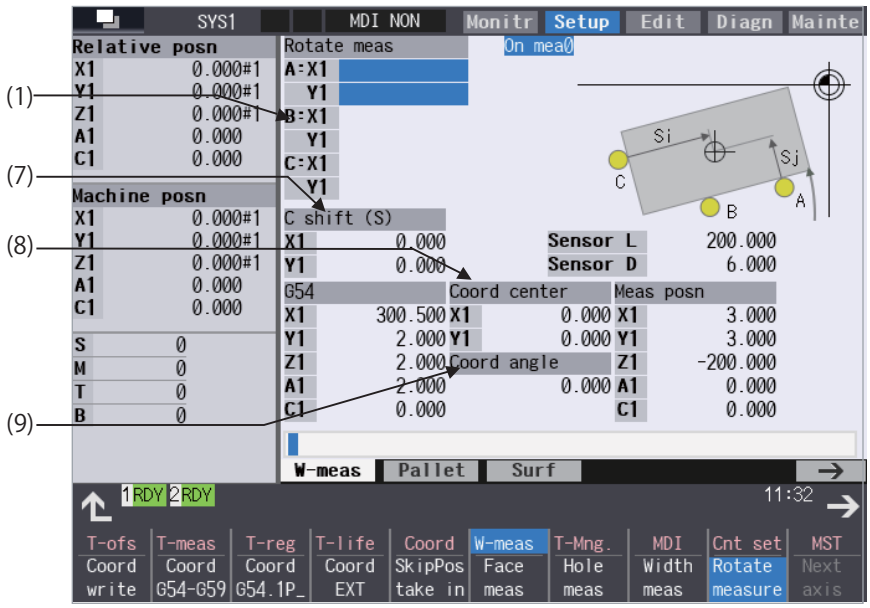
Setup – W-meas

The coordinate point for each axis can be measured on the workpiece measurement screen of the Setup screen. Install the sensor on the spindle, turn the measurement switch ON, and select manual feeding or handle feeding on the workpiece position measurement screen to bring the sensor into contact with the workpiece. Then, measure the coordinate point of each axis.
To make surface, hole, and width measurements, calculate each center coordinate from the measured coordinate, and set the obtained value to the workpiece coordinate system offset.
For the rotation measurement, the measurement result is set in the workpiece coordinate system offset (rotation center) and parameters "#8623 Coord rot centr(H)", "#8624 Coord rot centr(V)", and "#8627 Coord rot angle".

■For surface measurement (As for the hole or width measurement, the guide drawing (3) differs.)



■For rotation measurement



Manual numerical value command can be executed when manual numerical value command window is displayed by inputting an address key such as MST, etc.

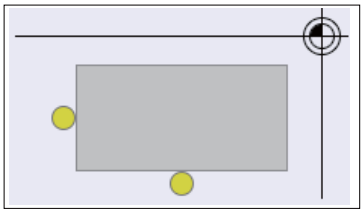
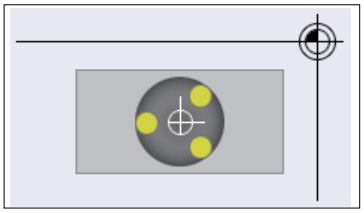
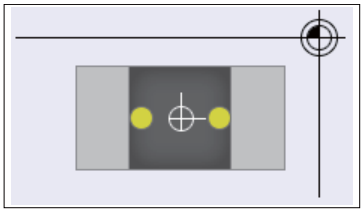
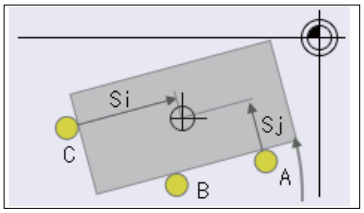
Display items

Display items		Details
(1)	Measurement counter	Displays the measurement position. As for the axis names, the 2nd axis names ("#1022 axname2") based on the basic axis name set by the parameters "#1026 base_I", "#1027 base_J" and "#1028 base_K" are displayed. 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 the menu [Face meas], [Hole meas], [Width meas], or [Rotate measure] is selected). The display contents differ depending on the measurement pattern (surface measurement, hole measurement, width measurement, or rotation measurement).
	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.
	Width measurement counter	Displays 2 (A, B) measurement positions.
	Rotation measurement counter	Displays 3 (A, B, C) measurement positions (X-, Y-axes).
(2)	Display of measurement in progress	"On mea" is displayed when the measurement switch on the machine operation board is ON. Refer to "Manual measurement status display" in "6.3.1 Tool Measurement (M System)" for details.
(3)	Guide graphic	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	Sensor length: Displays the length to the tip of the touch probe. ("#8701 Tool Length" setting value) Sensor diameter: Displays the diameter of the ball at the tip of the touch probe. ("#8702 Tool Dia" setting value)
(5)	Coordinate system offset	Displays the currently selected coordinate system offset.
(6)	Measurement position counter	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 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

Menus	Details
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> ♦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.) ♦When the parameter "#8924 Meas. confirm msg" is "1", this menu is highlighted and the operation 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 highlighting is canceled and the operation message is erased.
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 [Coord write].)
Coord G54.1 P_	Input the P number to display the selected extended workpiece coordinate system (G54.1 Pn) offset in the coordinate system offset section. (Perform the offset amount setting at the menu [Coord write].) This menu is not displayed if the extended workpiece coordinate system offset specification is disabled.
Coord EXT	This displays the external workpiece coordinate system offset in the coordinate system offset section. (Perform this setting at the menu [Coord write].)
SkipPos take in	This creates a false signal when performing simple measurement (measurement without using the touch probe) 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.
Face meas	This enables surface measurement. Surface measurement is possible when the power is turned ON.
Hole meas	This enables hole measurement. (The cursor moves to the measurement counter point A.)
Width meas	This enables width measurement. (The cursor moves to the measurement counter point A.)
Rotate measure	This enables rotation measurement. (The cursor moves to the measurement counter point A.)
Next axis	This changes the axes displayed at the coordinate system offset and measurement position counter every 5 axes. <Note> ♦This displays when the number of enabled axes is 6 or over.
Center shift	This sets the center shift amount. (The cursor moves to the 1st axis of "C shift (S)" when this is selected.)
Rotate center	This sets the coordinate rotation center. (The cursor moves to the 1st axis of "Coord center" when this is selected.)
Rotate angle	This sets the coordinate rotation angle. (The cursor moves to "Coord rot angle" when this is selected.)

Details of the "Measurement counter" and "Guide drawing" display areas

Menus	Measurement counter	Guide graphic
Face meas	Face meas X1 310.000 Y1 383.000 Z1 -9.900	
Hole meas	Hole meas A:X1 307.000 Y1 380.000 B:X1 276.553 Y1 342.009 C:X1 267.039 Y1 330.287	
Width meas	Width meas A:X1 17.354 Y1 17.354 Z1 -9.700 B:X1 34.878 Y1 41.035 Z1 -9.700	
Rotate measure	Rotate meas A:X1 117.472 Y1 105.000 B:X1 126.590 Y1 105.000 C:X1 205.000 Y1 122.209	

6.7.1.1 Carrying Out Surface Measurement

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 touch probe>

Measurement counter X = X axis skip position + sensor radius (*1)

Measurement counter Y = Y axis skip position + sensor radius (*1)

Measurement counter Z = Z axis skip position - sensor length

(*1) 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 probe)>

Measurement counter X: X axis measurement position

Measurement counter Y: Y axis measurement position

Measurement counter Z: Z axis measurement position

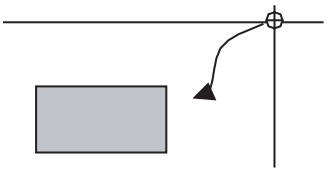
Operation method

(1) Press the menu [Face meas].

(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.

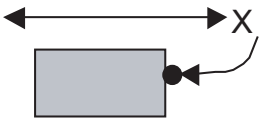
(4) Move the tool near the workpiece using manual feed and manual handle feed.



<Measurement using touch probe>

(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.



<Simple measurement (without the touch probe)>

(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) To select G55
Press the menus [Coord G54-G59] [G55].

(7) Press the menu [Coord write].

The value achieved by adding or subtracting the external workpiece coordinate 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.

6.7.1.2 Carrying Out Hole Measurement

For hole measurement, measure the measurement position coordinate of two axes (X and Y) at three points, calculate the center of the hole, and set the result to the designated workpiece coordinate offset. The measurement position coordinate is displayed on the measurement counter.

<Measurement using touch probe>

Measurement counter X = X axis skip position

Measurement counter Y = Y axis skip position

<Simple measurement (measurement without using touch probe)>

Measurement counter X = X axis machine position + center compensation (H) + skip past amount (Horizontal axis) (*1)

Measurement counter Y = Y axis machine position + center compensation (V) + skip past amount (Vertical axis) (*1)

(*1) 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 (H)	"#8707 Skip past amout(H)"
Skip past amount (V)	"#8708 Skip past amout(V)"

Operation method

(1) Press the menu [Hole meas]. → The menu is highlighted and the cursor displays at point A.
Use the [↑] or [↓] key to move to points B and C.

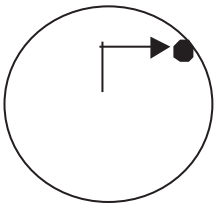
(2) Perform the reference position return command, etc. to position the tool at the basic point.

(3) Turn ON the measurement switch on the machine operation board.

(4) Move the tool into the hole using manual feed and manual handle feed.

<Measurement using touch probe>

(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 workpiece 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 probe (simple measurement)>

(5) Move the tool to the arbitrary position and press the menu [SkipPos take in]. → The skip position is calculated and the result is displayed for X and Y axes in the measurement counter A. 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) To select G55
Press the menus [Coord G54-G59] [G55].

(8) Press the menu [Coord write]. → The hole center position is measured, and based on the "#8709 Ext work sign rvs", value achieved by adding or subtracting the external workpiece coordinate offset 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.

6.7.1.3 Carrying Out Width Measurement

With width measurement, two points of the measurement position coordinate for each axis are measured, and then calculates the center between the points to set it as the workpiece coordinate offset. The measurement position displays at the measurement counter.

<Measurement using touch probe>
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 probe)>
Measurement counter X = X axis machine position + center compensation (H) + skip past amount (Horizontal axis) (*1)
Measurement counter Y = Y axis machine position + center compensation (V) + skip past amount (Vertical axis) (*1)
Measurement counter Z = Z axis machine position - sensor length

(*1) 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 (H)	"#8707 Skip past amout(H)"
Skip past amount (V)	"#8708 Skip past amout(V)"

Operation method

(1) Press the menu [Width meas].

The menu is highlighted and the cursor displays at point A.
Use the [↑] or [↓] key to move to points A and B.

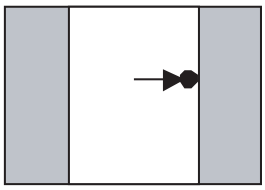
(2) Perform the reference position return command, etc. to position the tool at the basic point.

(3) Turn ON the measurement switch on the machine operation board.

(4) Move the tool to the center of the groove using manual feed or manual handle feed.

<Measurement using touch probe>

(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 workpiece 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.

<Simple measurement (without the touch probe)>

(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 movement axis section of the measurement counter A point.
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 workpiece 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) To select G55
Press the menus [Coord G54-G59] [G55].

(8) Press the menu [Coord write].

The width center position is measured, and based on the "#8709 Ext work sign rvs", value achieved by adding or subtracting the external workpiece coordinate 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.

6.7.1.4 Carrying Out Rotation Measurement

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 probe>
Measurement counter X = X axis skip position
Measurement counter Y = Y axis skip position

<Simple measurement (measurement without using touch probe)>
Measurement counter X = X axis machine position + center compensation (H) + skip past amount (Horizontal axis) (*1)
Measurement counter Y = Y axis machine position + center compensation (V) + skip past amount (Vertical axis) (*1)

(*1) 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 (H)	"#8707 Skip past amout(H)"
Skip past amount (V)	"#8708 Skip past amout(V)"

Operation method (Setting the center shift amount)

(1) Press the menu [Rotate measure].

(2) Press the menu [Center shift].

(3) Input the center shift amount (Si).
10.000 [INPUT]

(4) Input the center shift amount (Sj).
-5.000 [INPUT]

(5) Press the menu [Center shift].

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 [Rotate measure].

(2) Press the menu [Rotate center].

(3) Input data in the 1st axis of "Coord center" column.
0 [INPUT]

(4) Input data in the 2nd axis of "Coord center" column.
0 [INPUT]

(5) Press the menu [Rotate center].

Note

(1) Set the angle of the coordinate rotation in the same manner.

Operation method (Carrying out the rotation measurement using touch probe to set the measurement results to the workpiece coordinate offset)

(1) Press the menu [Rotate measure]. → The menu is highlighted and the cursor displays at point A.
The cursor can be moved to A, B, or C, using the [↑] or [↓] key.

(2) Set the parameter "#8116 Coord rot para invd" to "1" (invalid).

(3) Turn the manual absolute switch ON.

(4) Set the following parameters to "0".
#8623 Coord rot centr(H)
#8624 Coord rot centr(V)
#8627 Coord rot angle (*1)

(5) Perform the reference position return command, etc. to position the tool at the basic point.

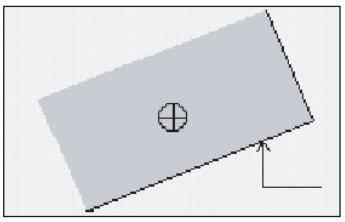
(6) Turn ON the measurement switch on the machine operation board.

(7) Referring to "Setting the center shift amount", set the center shift amount.
This setting is not necessary if the coordinate rotation center is served as the workpiece coordinate zero point.

(*1) 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.

(8) <Measurement using touch probe>

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.

<Without the touch probe (simple measurement)>

Move the spindle to the arbitrary position and press the menu [SkipPos take in].

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 menus [Coord G54-G59] [G55].

(11) Press the menu [Coord write].

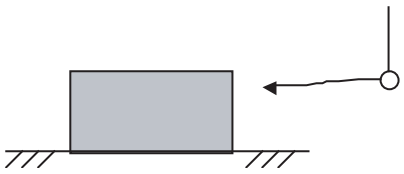
The hole center position is measured.
Based on the parameter "#8709 Ext work sign rvs" setting, the value achieved by adding or subtracting the external workpiece coordinate offset to or from the measurement result is set in workpiece coordinate system selected at (10).
The parameters "#8623 Coord rot centr(H)", "#8624 Coord rot centr(V)", and "#8627 Coord rot angle" are updated.
The measurement counter displays blank.

(12) Return the tool to the reference position, and turn the measurement switch OFF.

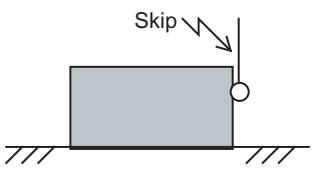
6.7.1.5 Performing Automatic Recontact When Contacting the Workpiece

When performing workpiece position measurement in either the JOG or handle mode, the following operations (automatic recontact) are performed after contacting the workpiece.

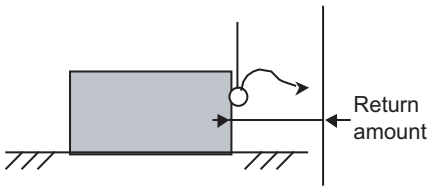
(1) Moves to the measurement point using either the JOG or handle mode.



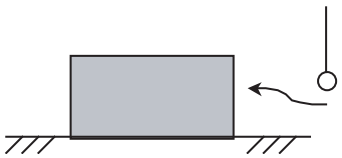
(2) Stops at the measurement point.



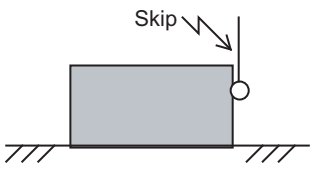
(3) Returns by the measurement parameter return amount (#8705).



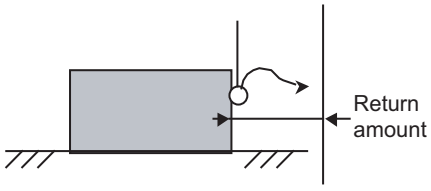
(4) Moves to measurement point again at the measurement parameter feedrate (#8706).



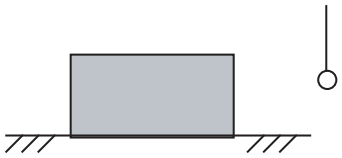
(5) Stops at the measurement point. (The skip position is read.)



(6) Returns by the measurement parameter return amount (#8705).



(7) The next measurement can be performed after automatically contacting the workpiece again.



The return speed for (3) and (6) is 40 times the measurement parameter feedrate (#8706). However, if the return speed is faster than rapid traverse rate (override 100%), the rapid traverse rate will become the return speed.

6.7.2 Workpiece Measurement (L System)

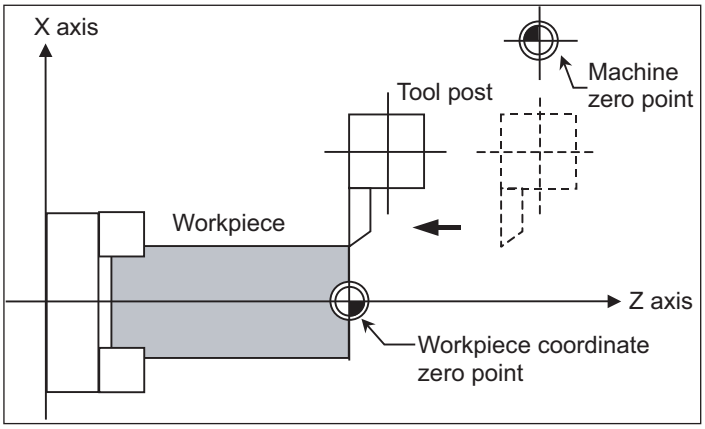
The external workpiece coordinate offset data for the Z axis (the 2nd axis) can be set by cutting the workpiece face by means of manual operations and inputting the workpiece measurement signal. By pressing the menu key, data can be set in the Z axis of an arbitrary coordinate offset.
Coordinate offset setting for the Z axis (the 2nd axis) of the part system 1 to 8 is possible.
Note that the workpiece coordinate offsets for multiple part systems cannot be measured at the same time.

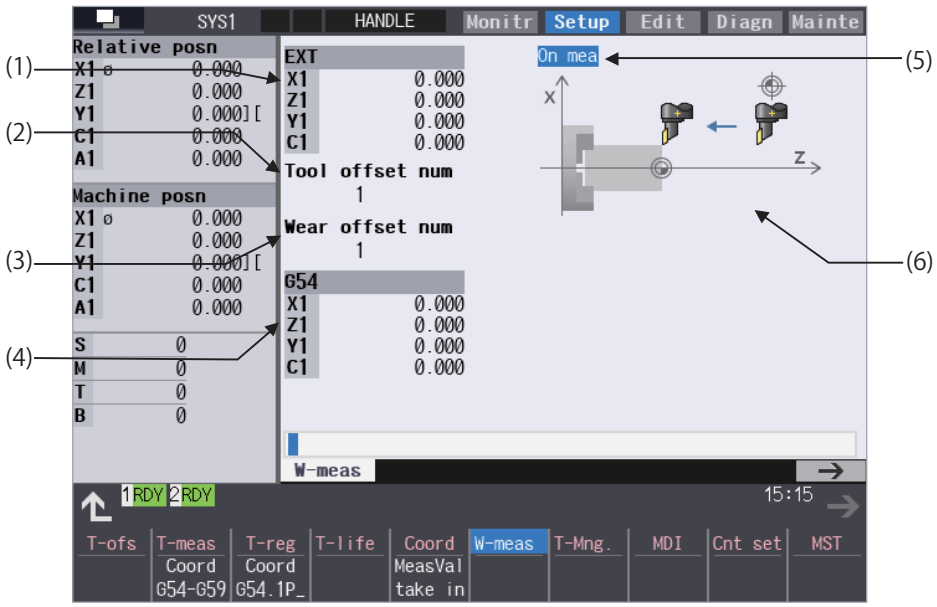
The workpiece measurement screen is displayed when the parameter "#11056 Workshift invalid" is "1". The parameter is set by MTB.
For signals and R registers, refer to the instruction manual issued by the MTB.

Note

(1) 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 items		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 L offset num	This displays the compensation No. of tool length data used by automatic calculation. <Note> ・If an illegal value is set, the value is blank. ("0" is displayed.)
(3)	Wear offset num	This displays the compensation No. of tool nose wear data used by automatic calculation. <Note> ・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.)
(4)	Coordinate system offset	This displays the coordinate system offset in which the measurement result is written when the menu [MeasVal take in] is pressed. G54 is displayed in default. When the menu [MeasVal take in] is pressed, the measurement result is set to 2nd axis of the external workpiece coordinate. <Note> ・The 5th axis and following are not displayed. ・The display of coordinate system offset area is switched by page up/down keys. For details, refer to "6.8.4 Changing the Coordinate System Display".
(5)	"On mea" display	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

Menus	Details
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.
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 extended workpiece coordinate system offset specification is invalid, this menu is grayed out and non-selectable.
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, the external workpiece coordinate offset and the workpiece base point offset. The calculated data is stored in Z axis (the 2nd axis) of the selected coordinate system offset.

Operation method (When the workpiece measurement signal is used)

Refer to the manual issued by MTB for details on each switch described in the operation procedures below. PLC devices for the signal of 1st part system are used for explanation.

(1) Execute the reference position return.
If the workpiece is measured without establishing the reference position return, the message "Meas axis not returned to ref. position" appears.

→ Carry out the dog-type reference position return, and establish the coordinate system.
In the absolute position detection specification, configure the initial setting in advance if the absolute position is not established.

(2) Set the mode selection switch to the manual mode (handle or jog).

(3) Turn ON the tool measurement mode signal (YC21).

→ "On mea" is displayed on the workpiece measurement screen of the setup screen.

(4) Select whether to measure the workpiece coordinate system shift value on the main spindle side or sub spindle side. Use the tool presetter sub-side valid signal (YCDA) to select main side or sub side.

<Note>

- Do not change status of the switch until the measurement is completed.

(5) Issue the T command by MDI operation, etc., then select tool.
Set the length data and wear data for the tool to be used beforehand.

→ (For MTB)
Set the selected tool compensation No. in R register.
The set R register differs depending on parameter setting and the state of the tool presetter sub-side valid signal.

(6) Cut the workpiece face.
This point is the program zero-point.

<Note>

- Do not move the tool in the Z axis (the 2nd axis) direction after cutting the workpiece face.
- If the workpiece does not need to be cut, carry out positioning to the measurement position.

(7) Display the workpiece measurement screen, and turn ON the workpiece coordinate measurement signal (Y9E1).

→ The Z axis (the 2nd axis) workpiece coordinate system offset data is automatically calculated from the machine coordinate at the point when the workpiece measurement signal is input, and the tool compensation data of the used tool. And the calculated value is set.

(8) Turn OFF the tool measurement mode signal (YC21).

[Details on formula]

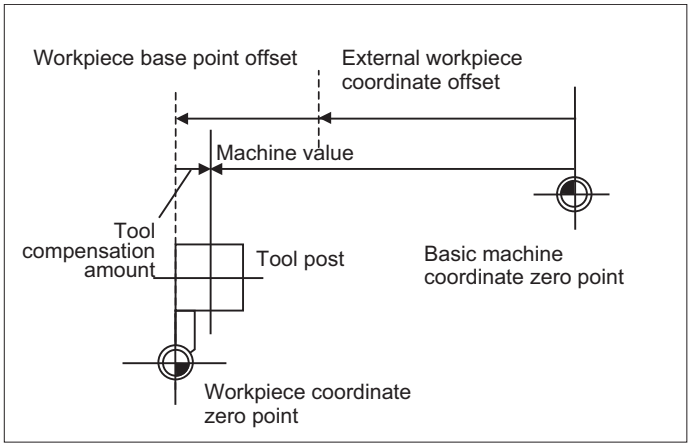
The external workpiece coordinate offset data is automatically calculated with the following expression.

External workpiece coordinate offset = Machine coordinate value - Tool compensation data - Workpiece base point offset

The tool compensation data used for the measurement is selected with the parameter "#1226 aux10// bit0".

#1226 aux10/bit0	Tool compensation data
0	Tool length data + Nose wear data
1	Tool length data

The workpiece base point offset used for the measurement is set with the parameter "#2701 Work base ofs".



[Compensation No. of selected tool]

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 Tlno.	#1130 set_t	#1218 aux02/ bit4	Tool L offset num		Wear offset num	
			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 tool compensation amount will be calculated as "0".
(2) If the compensation No. exceeds the number of specified offset sets, the message "Offset No. not found" appears.
(3) If the input data exceeds the setting range, the message "Data range error" appears.
(4) If the data protection is valid, the message "Data protect" appears.
(5) Set the compensation No. in R register by the BCD code.
(6) When the workpiece measurement screen is not displayed, the message will not appear even if the error occurs.
(7) The sign of the external workpiece coordinate offset varies depending on the parameter "#8709 Ext work sign rvs".

Operation method (When the menu [MeasVal take in] is used)

Refer to the manual issued by MTB for details on each switch described in the operation procedures below. PLC devices for the signal of 1st part system are used for explanation.

(1) Execute the reference position return.
If the workpiece is measured without establishing the reference position return, the message "Meas axis not returned to ref. position" appears.

→ Carry out the dog-type reference position return, and establish the coordinate system.
In the absolute position detection specification, configure the initial setting in advance if the absolute position is not established.

(2) Set the mode selection switch to the manual mode (handle or jog).

(3) Display the workpiece measurement screen by the [Setup] - [W-meas] menu, press the sub menu [Coord G54-G59] or [Coord G54.1P], and select the workpiece coordinate system to set the measurement value.

(4) Select whether to measure the workpiece coordinate system shift value on the main spindle side or sub spindle side. Use the tool presetter sub-side valid signal (YCDA) to select main side or sub side.
<Note>
- Do not change status of the switch until the measurement is completed.

(5) Issue the T command by MDI operation, etc., then select tool.
Set the length data and wear data for the tool to be used beforehand.

(For MTB)
Set the selected tool compensation No. in R register.
The set R register differs depending on parameter setting and the state of the tool presetter sub-side valid signal.

(6) Cut the workpiece face.
This point is the program zero-point.
<Note>
- Do not move the tool in the Z axis (the 2nd axis) direction after cutting the workpiece face.
- If the workpiece does not need to be cut, carry out positioning to the measurement position.

(7) Display the workpiece measurement screen, 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 offset data is automatically calculated from the machine coordinate at the point when the key is input, the tool compensation data of the used tool and the external workpiece coordinate offset value. And the calculated value is set in the Z axis (the 2nd axis) arbitrary workpiece coordinate offset.

[Details on formula]

The workpiece coordinate offset data is automatically calculated with the following expression.
The calculation formula changes depending on the parameter "#8709 Ext work sign rvs".

When "#8709" is "0":

Workpiece coordinate offset = Machine coordinate value - External workpiece coordinate offset - Tool compensation data - Workpiece base point offset

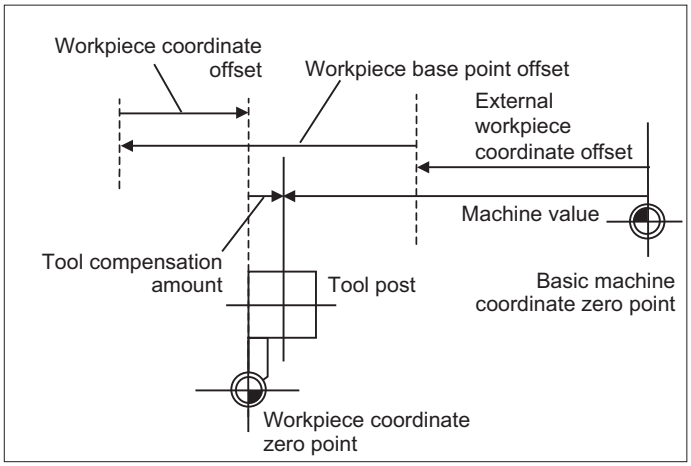
When "#8709" is "1":

Workpiece coordinate offset = Machine coordinate value+ External workpiece coordinate offset - Tool compensation data - Workpiece base point offset

The tool compensation data used for the measurement is selected with the parameter "#1226 aux10// bit0".

#1226 aux10/bit0	Tool compensation data
0	Tool length data + Nose wear data
1	Tool length data

The workpiece base point offset used for the measurement is set with the parameter "#2701 Work base ofs".



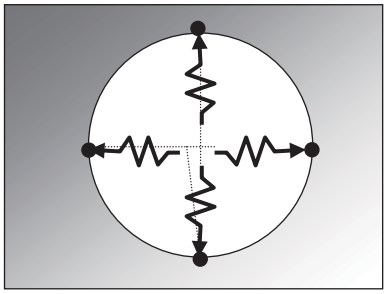
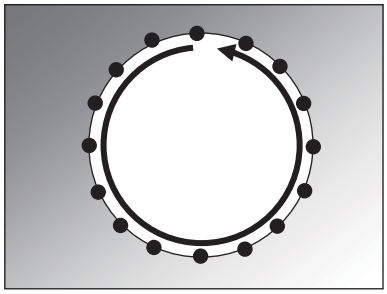
The selected tool compensation No. is the same as the measurement using the workpiece measurement signal. Refer to "Operation method (When the workpiece measurement signal is used)".

6.7.3 Measuring Instrument I/F Function

This function is to establish a system such as an on-machine measurement system by connecting NC and a measuring instrument.
This function allows you to establish the SPRINT system.

SPRINT is the sensor system manufactured by Renishaw, which measures the workpiece surface with a probe continuously at high speed. Compared with measurement with a touch probe, the measurement can be performed at higher speed and with higher accuracy.

For example of measuring a round shape hole, when a touch probe is used, four points (or three points) shown in (A) below are measured, and the radius, center, etc. of the circle are calculated based on the measurement points. On the other hand, when SPRINT is used, the hole side face is traced with a probe at high speed as shown in (B) below, the continuous coordinate values are read and the SPRINT application analyzes the measurement points and calculates the circle specifications.

(A) Measure a circle with a touch probe	(B) Measure a circle with SPRINT
	

This function is available on M800VW Series with the measuring instrument I/F specification as an option.
To establish the SPRINT system, the following items needs to be prepared by MTB.

- Function expansion unit FCU8-EX543
- Windows-based display unit (PC display unit) or industrial computer (IPC)
- Sensor system of Renishaw
- Cable to connect the SPRINT interface unit and NC

Operation method

Enable the SPRINT system, mount the SPRINT analog probe to the spindle, use the measurement software provided by Renishaw to create a measurement program, and then perform measurement.
For the SPRINT system screen, operation procedure and operation sequence, refer to the SPRINT system manual of Renishaw.

6.8 Setting Workpiece Coordinate System Offset

Setup – Coord

Monitr – Coord

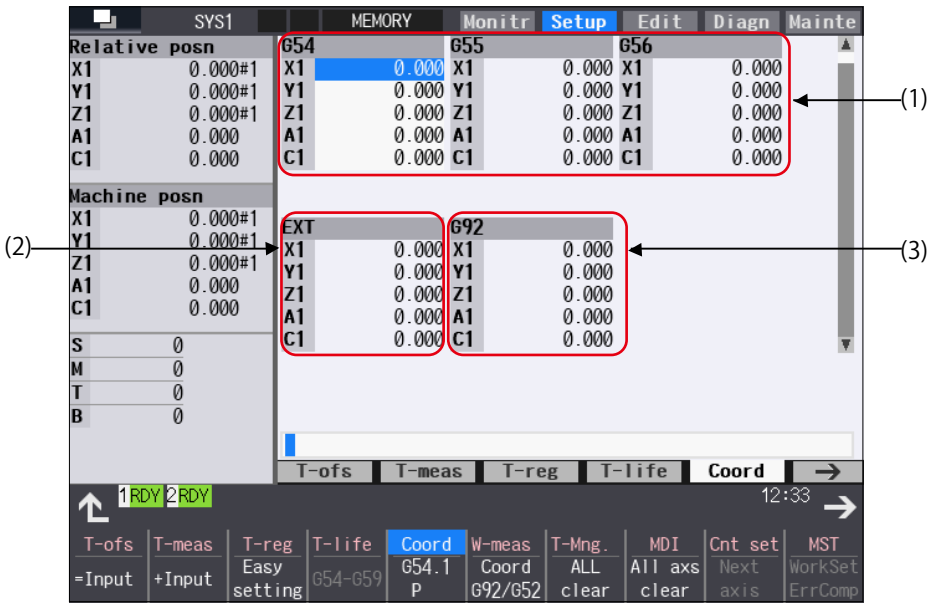
The coordinate system offset and workpiece installation error compensation controlled by the NC can be set and displayed on the monitor (Monitr) screen or the Setup screen.

Depending on the specifications, the number of coordinate system offset sets can be increased by 48, 96, or 300 sets.

Note

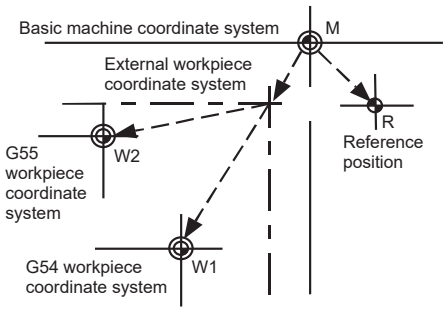
(1) The G92/G52 coordinate system offset cannot be set.

(2) Workpiece installation error compensation is displayed only when the workpiece installation error compensation specification is valid.



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 items		Details	
(1)	Coordinate system offset area	This sets and displays the offset amount for the workpiece coordinate system (G54 - G59), or the extended workpiece coordinate system (G54.1Pn). Use the menu operation or page up/down keys to specify which workpiece coordinate system offset to display or set. The workpiece coordinate system offset data can be set in absolute, incremental, or subtracted position. Setting range: - 99999.999 to 99999.999 (mm) (When "#1003 iunit"= B, micron)	
(2)	EXT offset area	This displays or sets the offset data for the external workpiece system.	
(3)	G92/G52 offset area	This displays the shift amount for G92 coordinate system or the offset amount for 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 the next block or after several subsequent blocks.

Menus

Menus	Details
=Input	This executes an absolute input. When "#8929 Disable=INPUT:comp" is set to "1", this menu cannot be selected. This menu is displayed when you select the menu [Input =/+] or [Input =/-] under the menu [Input menu].
+Input	This executes an incremental input. This menu is displayed when you select the menu [Input =/+] or [Input -/+] under the menu [Input menu].
-Input	This executes a subtraction input. This menu is displayed when you select the menu [Input-/+] or [Input =/-] under the menu [Input menu].
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. When "#8935 W COORD CONFIRM" is set to "1", a operation message is displayed, and the menu is highlighted.
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.
G54.1 P	Enter the P number displayed at the screen to display the extended workpiece coordinate (G54.1 Pn) offset in the coordinate system offset area. This menu is not displayed if the extended workpiece coordinate system offset specification is disabled. If the local coordinate system offset (G52) displays in the G92/G52 area, the display changes to G92.
Coord G92/G52	This displays the shift amount for G92 coordinate system or the offset amount for G52 coordinate system. The G92 offset data is always displayed when the cursor is not at the G54 to G59 offset data.
All clear	Coordinate system offset All coordinate system offset amounts for all axes are erased with the exception of G92 and EXT. Workpiece installation error compensation This erases all of workpiece installation error compensation data except the common workpiece installation error compensation amount.

Menus	Details
All axs clear	Coordinate system offset This erases the local offset data for all axes corresponding to the offset data of the coordinate system where the cursor is located. Workpiece installation error compensation This erases the coordinate system compensation data where the cursor is located.
Next axis	This changes the axes displayed at the coordinate system offset and measurement position counter every five axes. <Note> ♦This displays when there are 6 or more axes.
WorkSet ErrComp	This displays workpiece installation error compensation screen. While the workpiece installation error is displayed, the menu will be kept highlighted. <Note> ♦It cannot be selected when the workpiece installation error compensation specification is not valid.
RotAxis WorkCmp	This displays the screen of workpiece position offset for rotary axis. While the workpiece position offset for rotary axis is being executed, the menu will be kept highlighted. <Note> ♦If the specification of workpiece position offset for rotary axis is not enabled, this menu is non-selectable.
Input menu	Press this menu and select input menus to be displayed between [=Input], [+Input] and [-Input].

Note

(1) The menus [Easy setting], [G54-G59], [G54.1P], [Coord G92/52] and [Next axis] are non-selectable on the screen of the workpiece position offset for rotary axis.

(2) The menus [Easy setting], [G54-G59], [G54.1P] and [Coord G92/52] are non-selectable on the screen of the workpiece position offset for rotary axis. The menu [Next axis] is also non-selectable when the number of axes is 6 or less.

Sub-menus of [Input menu].

Menus	Details
Input =/+	This displays the menus [=Input] and [+Input].
Input -/+	This displays the menus [-Input] and [+Input].
Input =/-	This displays the menus [=Input] and [-Input].

6.8.1 Setting the Coordinate System Offset Amount

Operation method (Setting the workpiece coordinate system G54 - G59 offset amount)

(1) Press the menu [G54-G59]. → 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 ▲ and PAGE ▼ page keys to change the coordinate system display.
Refer to "Using the page up/down keys (previous page: PAGE ▲ /next page: PAGE ▼) to change counter display" in "6.8.4 Changing the Coordinate System Display" for details.)

(3) Use the [↑] or [↓] key to move the cursor to the axis for which data is to be set.

(4) Set the offset amount.

122 [INPUT]

The value can also be designated by pressing the menu [=Input] instead of the [INPUT] key.

(5) Set other data using the same method.

Note

(1) It is possible to perform incremental input by pressing the menu [+Input] key instead of the [INPUT] key or the menu [=Input].
(Example) If the original data is 122.000:
1.234 [+Input] -> 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 "Chapter 12 Performing a Backup Operation" for details of the data transfer function.)

6.8.2 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 [All axs clear]. → A message displays confirming whether it is okay to erase the data.

(2) Press the [Y] 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 [All clear]. → A message displays confirming whether it is okay to erase the data.

(2) Press the [Y] 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

(1) G92 data cannot be erased.

6.8.3 Setting the Workpiece Coordinate Origin

Press the menu [Easy setting] 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 the axis where the cursor is located.

If the menu [Easy setting] 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.

Operation method

(1) Move the machine position by jog feed etc.

(2) Move the cursor to the axis of the coordinate system to be set using the [↑] or [↓] key.

(3) Press the menu [Easy setting]. → Current machine coordinates are set to the axis where the cursor is located as the workpiece coordinate zero point.

Note

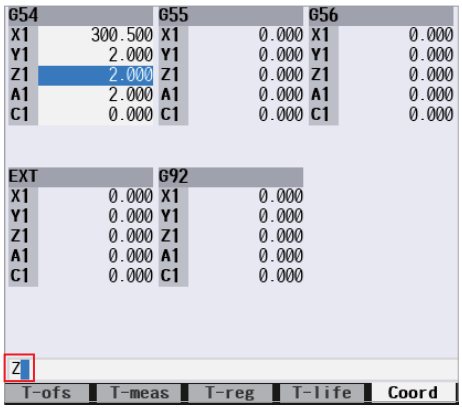
(1) When the parameter "#8935 W coord confirm" is set to "1", the menu [Easy setting] is highlighted and the confirmation message appears. Press the [Y] or [INPUT] key to complete the setting.

Operation method (Set the workpiece coordinate zero point by subtracting input value)

(1) Move the machine position by jog feed etc.

(2) Enter an axis name in input area. → The cursor moves to the input axis.

(Example) When you enter "Z" in the input area and the cursor moves to Z axis.



(3) Enter the value to be subtracted from the machine position next to the input axis name.

(4) Press the menu [Easy setting]. → "Machine position subtracted the input value" is set as the offset value of the axis which is selected in step (2).

(Example 1) 100.0 - 50.0 = 50.0

(Example 2) 100.0 - (-50.0) = 150.0

(Example 1)

To input the offset value "50" when the machine position of Z axis is "100.0"

Z50 [Easy setting]

(Example 2)

To input the offset value "-50" when the machine position of Z axis is "100.0"

Z-50 [Easy setting]

Note

(1) When the parameter "#8935 W coord confirm" is set to "1", the menu [Easy setting] is highlighted and the confirmation message appears. Press the [Y] or [INPUT] key to complete the setting.

(2) When an address code to open the manual numerical value command window which is set by the parameter "#1170 M2name" (Second miscellaneous code) is the same as the axis name set by the parameter "#1013 axname" (Axis name), the address code for the manual numerical value command window takes precedence. Calculation of "Machine position subtracted the input value" is not available.

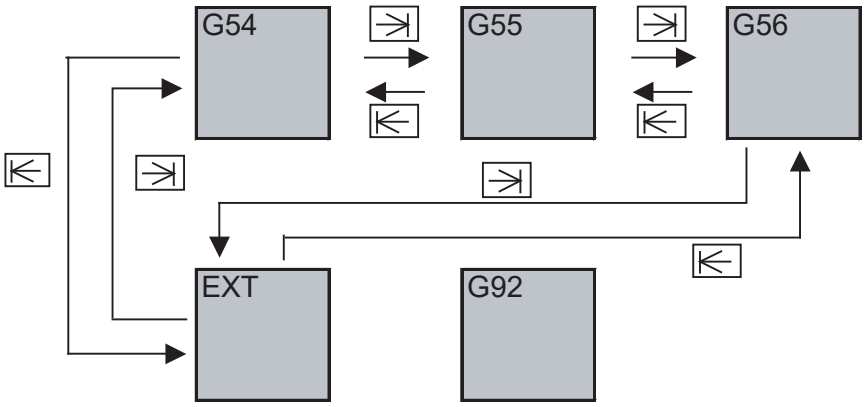
(3) When multiple axes of the same name are set by the parameter "#1013 axname", the first detected axis after the cursor position is highlighted.

(4) If you enter an illegal value in the input area, the message "Setting error" appears when the [Easy setting] menu is pressed.

6.8.4 Changing the Coordinate System Display

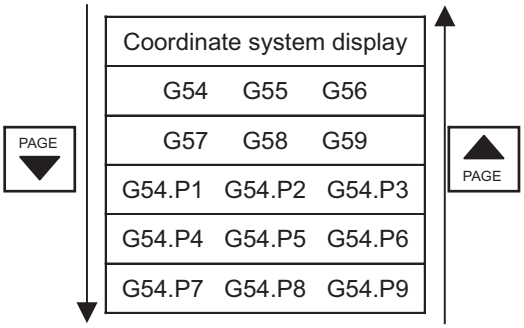
Using the tab keys (/) to change cursor position

The counter on which the cursor is displayed can be switched using the tab keys as shown below.



Using the page up/down keys (previous page: /next page:) to change counter display

The coordinate system offset display can be switched using the page up/down keys as shown below.



The optional specification is required for the G54.P1 - P300 workpiece coordinate system offset display.

6.8.5 Changing the Input Menus to Display

The offset data can also be input by menu operation, with absolute, incremental or subtracted position. You can change the menus to be displayed on the screen by pressing the menu [Input menu].

Operation method

(Example) To display the menus [-Input] and [+Input]

(1) Press the menu [Input menu]. → The menu which is currently being selected will be high-lighted.

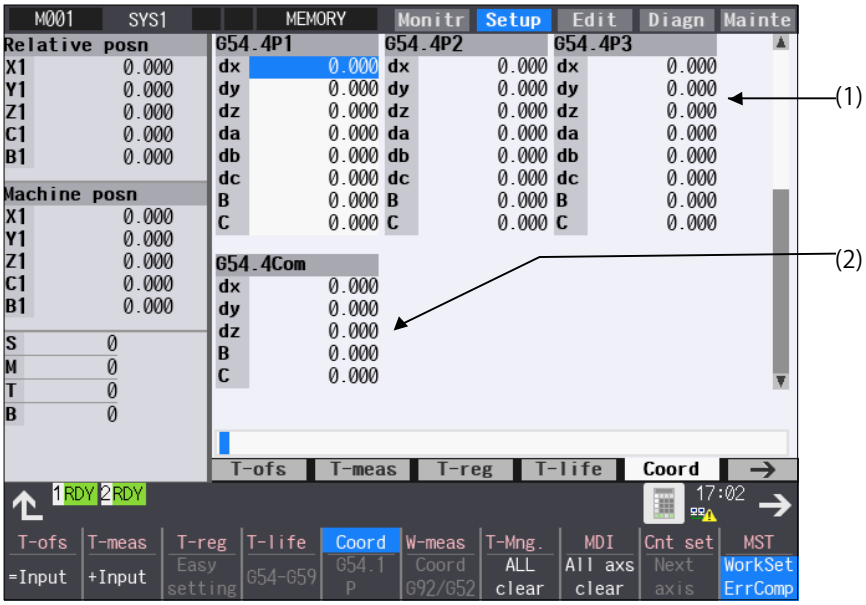
(2) Press the menu [-/+ Input]. → The menus [-Input] and [+Input] are displayed.

Note

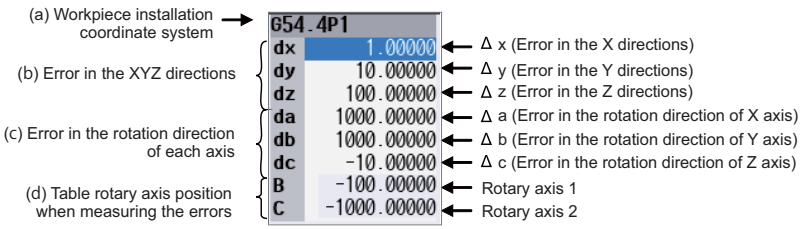
(1) The selected menus are maintained even after the power is turned ON again.

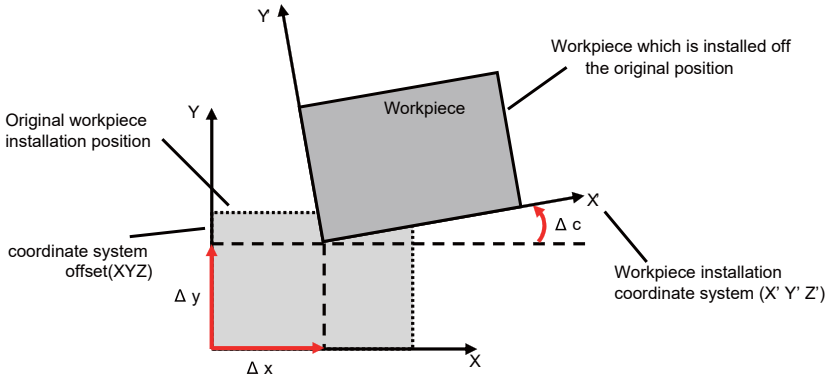
6.8.6 Setting the Offset Amount of Workpiece Installation Error

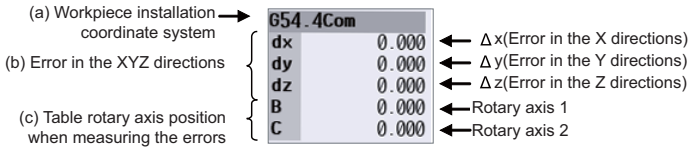
This function can set and display the workpiece installation error compensation controlled by the NC.
This function is an optional specification. And this is the function for each part system.



Display items

Display items		Details
(1)	Workpiece installation error compensation amount	This displays and sets an offset amount of the workpiece installation coordinate system (G54.4Pn). The workpiece coordinate system offset data can be set in absolute, incremental, or subtracted position. The number of display or setting digits is determined by the value set in "#1003 iunit" for $1. (Example) - 99999.999 to 99999.999 (mm) ("#1003 iunit"= B, micron) Details of displayed contents are as follows:  (a) Workpiece installation coordinate system Workpiece installation error displays seven sets of error amount (G54.4P1 to G54.4P7) corresponding to the compensation No. commanded with address P. (b) Errors in the XYZ directions (Δx, Δy, Δz) Set the zero point of the workpiece installation coordinate system seen from the original workpiece coordinate system. The error name of the each axis direction starts with d (fixed) then a small letter of an orthogonal coordinate axis (the axis name set in #7900 RCDAX_I, #7901 RCDAX_J, #7902 RCDAX_K) of $1. •#7900 RCDAX_I ... If X, it is "dx". •#7901 RCDAX_J ... If Y, it is "dy". •#7902 RCDAX_K ... If Z, it is "dz". <Note> •When the orthogonal coordinate axis (the axis name set in #7900 RCDAX_I, #7901 RCDAX_J, #7902 RCDAX_K) is two letters, only the first letter is added in lowercase letters and displayed.

<table>
<tr><th colspan="2">Display items</th><th>Details</th></tr>
<tr><td>(1)</td><td>Workpiece installation error compensation amount
(Continued)</td><td>(c) Error in the rotation direction of each axis (Δa, Δb, Δc)
Set Δa, Δb, and Δc to determine the XYZ directions of the workpiece installation coordinate system. Compared to the original workpiece coordinate system, the workpiece installation coordinate system's X axis is rotated by the angle Δa, Y axis is rotated by the angle Δb, and Z axis is rotated by the angle Δc.
The positive direction of the rotation angle is the right-handed rotation.
(d) Table rotary axis position when measuring the errors
Set the machine coordinate values of the table rotary axis when Δx, Δy, Δz, Δa, Δb, and Δc are measured.
The setting is not required for the machine configuration which does not have the table rotary axis.
Two axes should be set for 5-axis machine table tilt type.
Axis name to be displayed in the rotary axis 1 and 2 differs depending on the machine configuration.
♦Table tilt type
Displays the axis name #7942 ROTAXW1 for "Rotary axis1", the axis name #7952 ROTAXW2 for "Rotary axis 2"
♦Tool tilt type
The axis name "#7922 ROTAXT1" will be displayed for "rotary axis 1", and the axis name "#7932 ROTAXT2" will be displayed for "rotary axis 2".
♦Combined type
The axis name "#7932 ROTAXT2" will be displayed for "rotary axis 1", and the axis name "#7952 ROTAXW2" will be displayed for ,"rotary axis 2".


<Note>
♦XYZ are set for the basic axis IJK in this example.</td></tr>
<tr><td>(2)</td><td>Common workpiece installation error offset amount</td><td>You can set the error amount which is commonly added to the seven sets of the error amount corresponding to the compensation No. commanded with address P for the workpiece installation error (G54.4P1 to G54.4P7) in this area.
Details of displayed contents are as follows:


(a) Workpiece installation coordinate system
Displays the error amount which is commonly added to the seven sets of the error amount corresponding to the compensation No. commanded with address P for the workpiece installation error (G54.4P1 to G54.4P7).
(b) Errors in the XYZ directions (Δx, Δy, Δz)
Refer to the display item (1) "Workpiece installation error compensation amount".
(c) Table rotary axis position when measuring the errors
Refer to the display item (1) “Workpiece installation error compensation amount".
<Note>
♦Error in the rotation direction of each axis (Δa, Δb, Δc) cannot be set.</td></tr>
</table>

⚠ CAUTION

If the workpiece installation error compensation amount is changed during automatic operation (including during single block stop), the change will be valid from the command in the next block or after several subsequent blocks.

Menus

For details, refer to "6.8 Setting Workpiece Coordinate System Offset".

Setting the workpiece installation error data

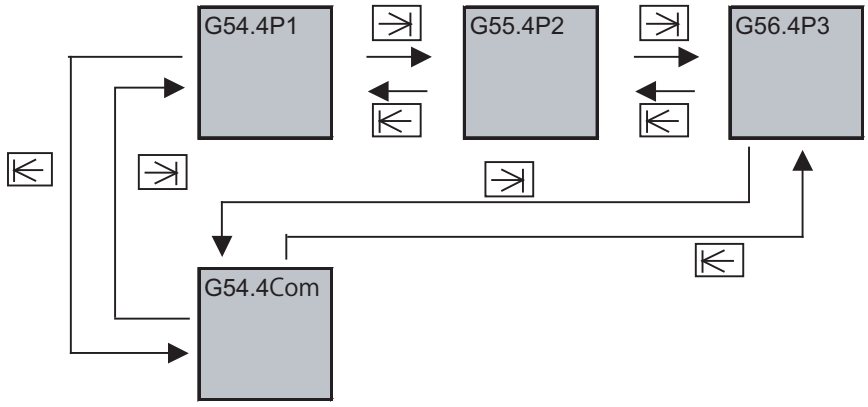
For details, refer to "6.8.1 Setting the Coordinate System Offset Amount".

Erasing the workpiece installation error data

For details, refer to "6.8.2 Erasing the Coordinate System Offset Amount".

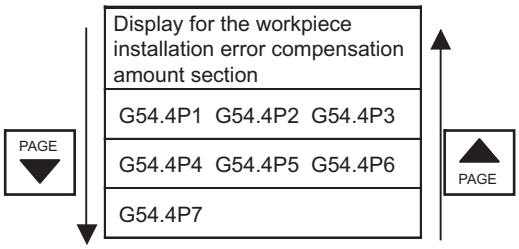
Changing the cursor position using tab keys (/)

The counter on which the cursor is displayed can be switched using the tab keys as shown below.



Changing the counter display using the page up/down keys (previous page: PAGE / next page: PAGE)

The counter display of the workpiece installation error compensation amount can be switched using the page up/down keys as shown below. The cursor moves to "dx" (the first line) when the counter is switched.



Changing the input menus to display

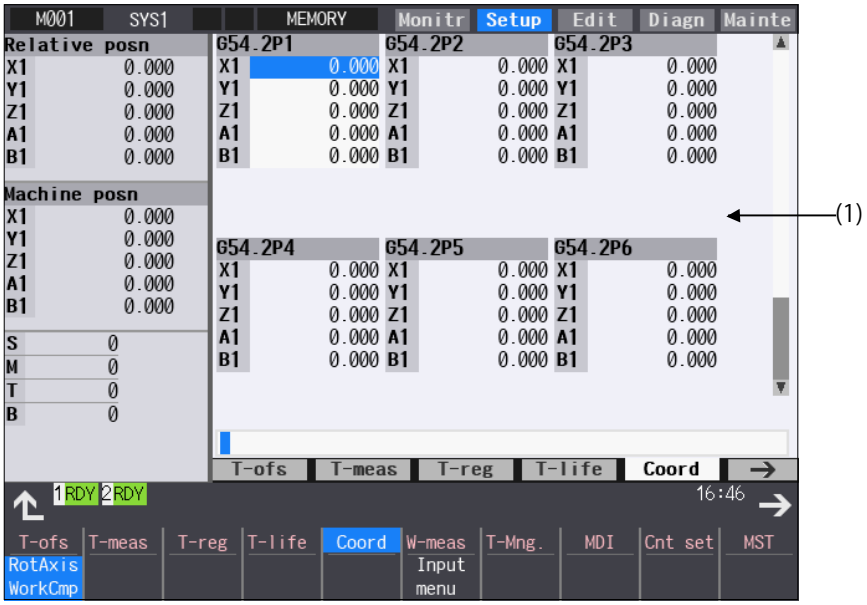
For details, refer to "6.8.5 Changing the Input Menus to Display".

6.8.7 Setting the Offset Amount of Workpiece Position for Rotary Axis

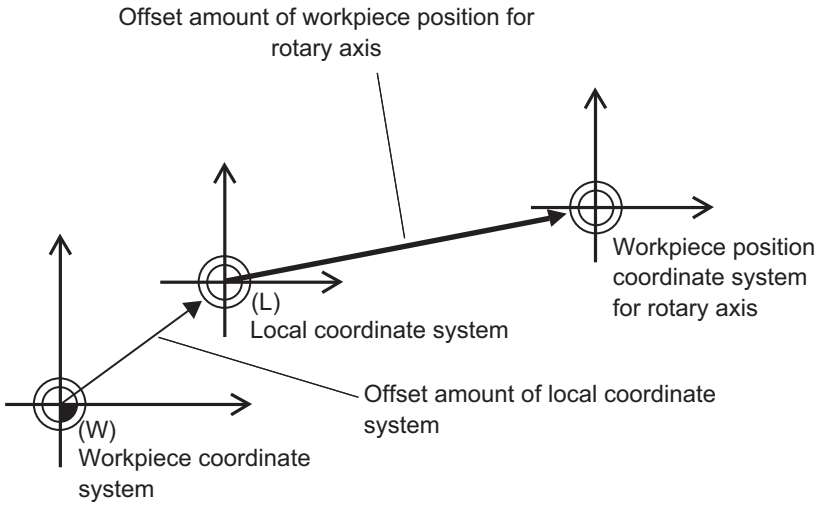
This function can set and display the workpiece position offset for rotary axis controlled by the NC.

This function is an optional specification. This function is available only in the first part system of M system.

Refer to "Programming Manual" for details on this function.



Display items

Display items		Details
(1)	Workpiece position offset for rotary axis	The workpiece position offset amount for rotary axis (G54.2 Pn) can be displayed and set. The workpiece position offset amount for rotary axis can be set in absolute, incremental, or subtracted position. Setting range: - 99999.999 to 99999.999 (mm) ("#1003 iunit"= B, micron) 

⚠ CAUTION

If the workpiece position offset amount for rotary axis is changed during automatic operation (including during single block stop), the change will be valid after the next G54.2 command is issued.

Menus

For details, refer to "6.8 Setting Workpiece Coordinate System Offset".

Setting the workpiece position offset for rotary axis

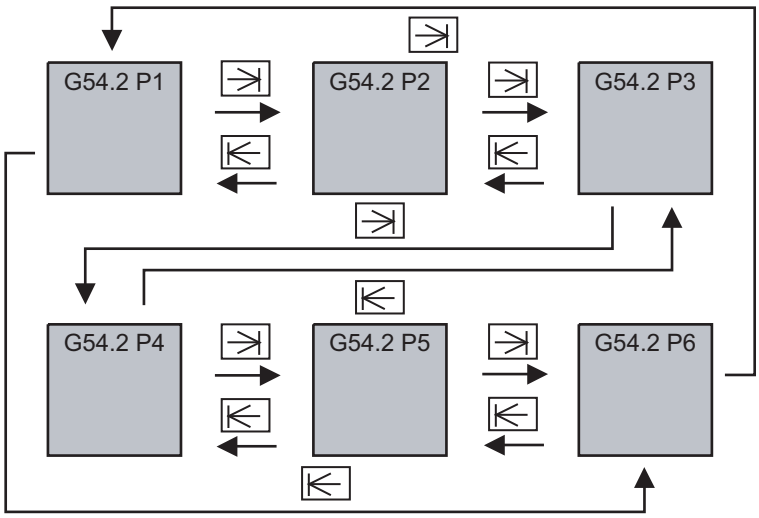
For details, refer to "6.8.1 Setting the Coordinate System Offset Amount".

Erasing the workpiece position offset for rotary axis

For details, refer to "6.8.2 Erasing the Coordinate System Offset Amount".

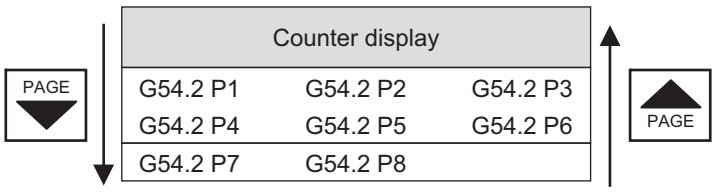
Changing the cursor position using tab keys (/)

The counter on which the cursor is displayed can be switched using the tab keys as shown below.



Changing the counter display using the page up/down keys (previous page: / next page:)

The counter display of the workpiece position offset amount can be switched using the page up/down keys as shown below.



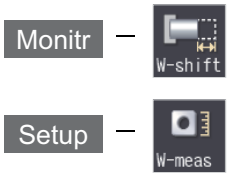
Changing the displayed axis

For the function of workpiece position offset for rotary axis, the number of axes that can be displayed simultaneously is 6. To display the subsequent axes, press the menu [Next axis].

Changing the input menus to display

For details, refer to "6.8.5 Changing the Input Menus to Display".

6.9 Setting Workpiece Coordinate System Shift (L System)



This function allows you to shift the workpiece coordinate system.
Workpiece coordinate system that is created at the machining program creation may not match to the coordinate system that actually programmed or programmed by automatic coordinate system setting. This function allows you to machine without changing the machining program by shifting from the coordinate system that is assumed when the program is created to the measured coordinate system. In addition, workpiece can be machined using the same program even if their shapes are changed.

The workpiece coordinate system shift value can be designated by manual setting or automatic setting.

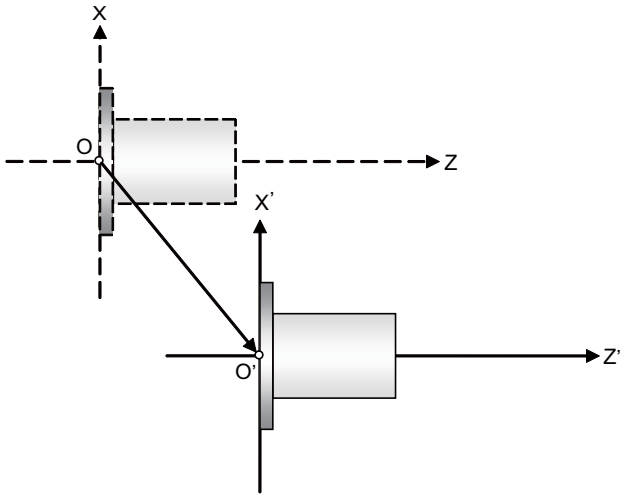
Manual setting: Allows you to directly input the workpiece coordinate system shift value from the workpiece coordinate system shift screen or select the menu [Measure] to designate the workpiece coordinate system shift value.

Automatic setting: Allows you to designate the workpiece coordinate system shift value with the workpiece measurement signal.

The manual setting and automatic setting to designate the workpiece coordinate system shift value using the menu [Measure] are optional specifications.

When this function is enabled, the menu [W-shift] is displayed on the monitor (Monitr) screen. If the menu [W-meas] is selected on the setup (Setup) screen, the setting screen for the workpiece coordinate system shift screen is displayed.

This function is enabled when the parameter "#11056 Workshift invalid" is "0". The parameter is set by MTB.



Set the shift value from O to O' as the workpiece coordinate system shift value.

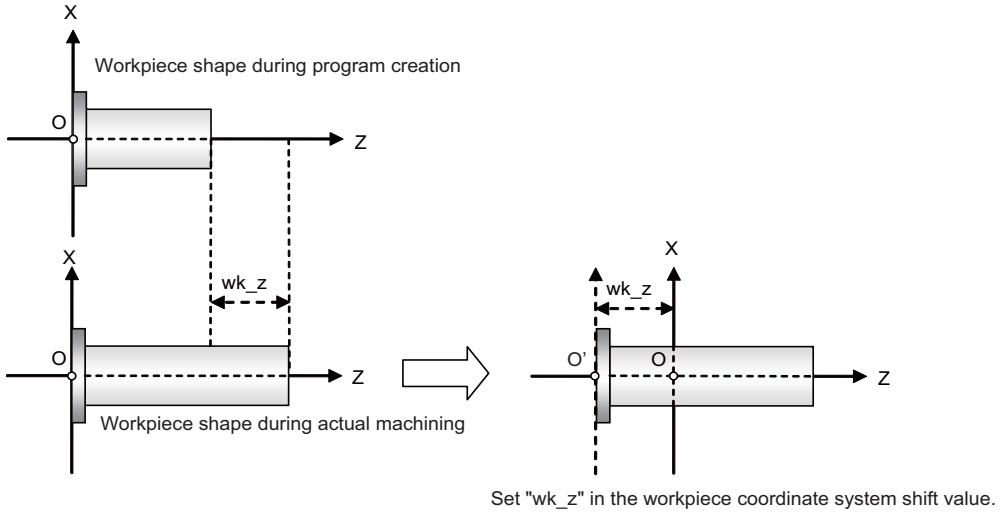
X, Z: Coordinate system established when the program is created

X', Z': Actually measured coordinate system

Note

(1) When this function is disabled, the menu [W-shift] is not displayed on the monitor (Monitr) screen. If the menu [W-meas] is selected on the setup (Setup) screen, the external workpiece setting screen is displayed.

[Example of using workpiece coordinate system shift]



X - Z: Coordinate system established when the program is created

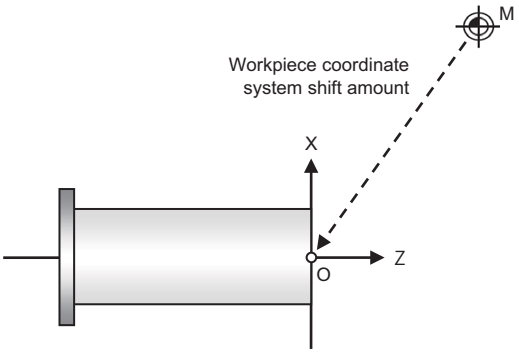
wk_z: The degree of change of the workpiece shape

[Workpiece base point offset]

In some machine configurations, it is difficult to see that from where the workpiece coordinate system shift amount is set. In such a case, offset the base point of the workpiece coordinate system shift amount to recognizable position, e.g. chuck center. That enables the shift amount to be set easily. The amount of workpiece base point offset is required to be set by MTB. The amount of workpiece base point offset is set to the parameter "#2701 Work base ofs".

The following explains details on using the workpiece base point offset.

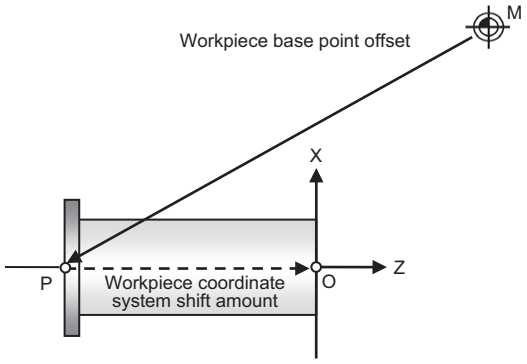
When the workpiece base point offset is not set



Set the distance from M to O as the workpiece coordinate system shift amount.

M: Machine coordinate zero point
O: Coordinate system zero point when the program is created

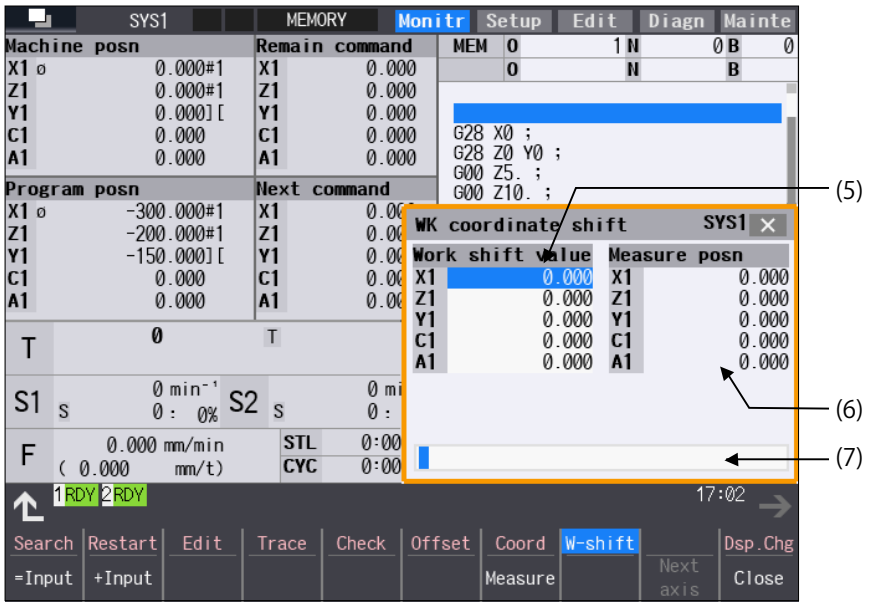
When the workpiece base point offset is set



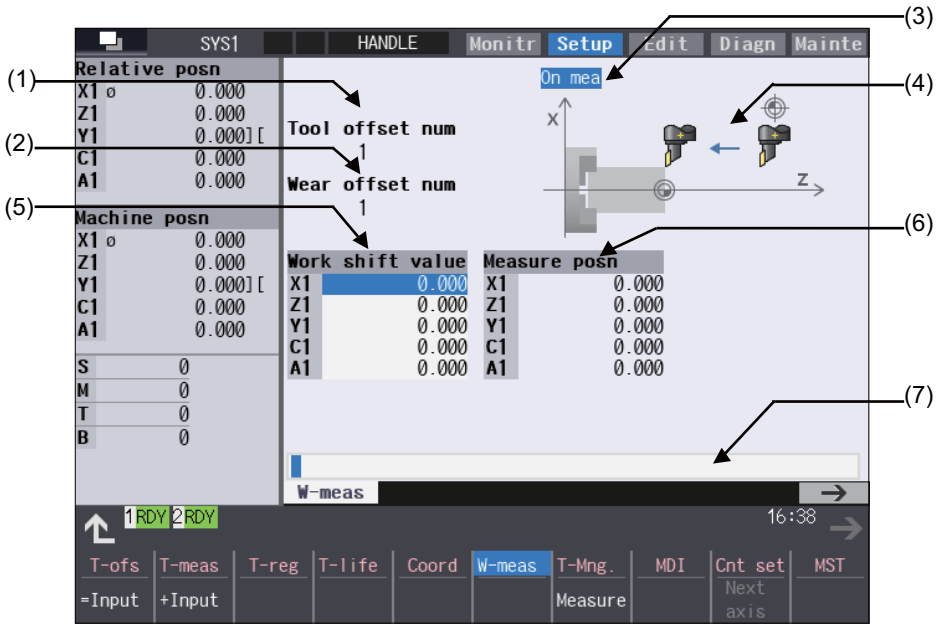
Set the distance from M to P as the workpiece base point offset amount so that the workpiece coordinate system shift amount can be set easily. Set the distance from P to O as the workpiece coordinate system shift amount.

M: Machine coordinate zero point
P: Base point of the workpiece coordinate system shift amount (arbitrary position)
O: Coordinate system zero point when the program is created

<Monitor screen>



<Setup screen>



Display items

	Display items	Details
(1)	Tool L offset num	This displays the compensation No. of the tool length data used in the automatic setting. (*1)
(2)	Wear offset num	This displays the compensation No. of the tool nose wear data used in the automatic setting. (*2)
(3)	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 coordinate system shift write mode signal. (*3)
(4)	Guide graphic	This displays the measurement area.

Display items	Details
(5) Work shift value	This displays the currently designated workpiece coordinate system shift value. (*4)(*5) The initial cursor position is located at the 1st axis in all part systems. If the cursor position is changed using cursor movement keys or the menu [Next axis], it is held until the program is placed into the following status. Monitor screen: The cursor position is not held. Setup screen: The cursor position is held until the power is turned on again.
(6) Meas posn	The measurement position (machine position - tool length compensation amount of the T-commanded tool) is displayed. (*4)(*5)
(7) Input section	This displays the input data.

(*1) If an illegal value is set, the value is blank ("0" is displayed).

(*2) If an illegal value is set or tool nose wear data is not used for measurement (parameter "#1226 aux10/bit0" is "1"), the value is blank ("0" is displayed).

(*3) When the workpiece coordinate offset measurement specification is invalid, the value is not displayed during measurement even if the measurement switch is set ON.

(*4) To display the 9th axis and after, use the menu [Next axis].

(*5) When a value out of the range of the workpiece shift amount which is set on the range setting screen, the message "Setting error" appears.

Menus

<Monitor screen>

Menus	Details
=Input	This executes an absolute input. When the parameter "#8929 Disable=INPUT:comp" is set to "1", this menu is disabled.
+Input	This executes an incremental input.
Measure	The workpiece coordinate system shift value is measured.
Next axis	This changes the workpiece shift value and the axes displayed at the measurement position on a 8-axis basis. The cursor of the workpiece shift value then jumps to the top row. <Note> ♦When nine or more axes are included in a part system, this menu is enabled.
Close	This closes the pop-up window and quits this function.

<Setup screen>

Menus	Details
=Input	This executes an absolute input. When the parameter "#8929 Disable=INPUT:comp" is set to "1", this menu is disabled.
+Input	This executes an incremental input.
Measure	The workpiece coordinate system shift value is measured.
Next axis	This changes the workpiece shift value and the axes displayed at the measurement position on a 8-axis basis. The cursor of the workpiece shift value then jumps to the top row. <Note> ♦When nine or more axes are included in a part system, this menu is enabled.

6.9.1 Manually Setting the Workpiece Coordinate System Shift Value

Operation method (Set the absolute position "10.000" to the workpiece coordinate system shift value of the Z axis (2nd axis) on the Monitor screen.)

(1) This displays the workpiece coordinate system shift.
<Monitor screen>
Press the menu [W-shift].
<Setup screen>
Press the menu [W-meas].
→ The workpiece coordinate system shift screen is displayed.

(2) Use the [↑] or [↓] key to move the cursor onto the Z axis.
→ The cursor moves to the Z axis.

(3) Designate the workpiece coordinate system shift value.
10 [INPUT]
The value can also be designated by pressing the menu [=Input] instead of the [INPUT] key.
→ The designated value appears.
The cursor moves to the next axis.

Note

(1) If the parameter "#11017 T-ofs set at run" (Tool compensation amount setting during automatic operation enabled) is set to "1", the workpiece coordinate system shift value can also be designated in the automatic operation mode or operation pause mode.

Operation method (Add the value "1.234" to the workpiece coordinate system shift value of the C axis (3rd axis) on the Monitor screen.)

(1) This displays the workpiece coordinate system shift.
<Monitor screen>
Press the menu [W-shift].
<Setup screen>
Press the menu [W-meas].
→ The workpiece coordinate system shift screen is displayed.

(2) Use the [↑] or [↓] key to move the cursor onto the C axis.

(3) Input a numeric value.
1.234 [+Input]
→ The input value added to the original value is displayed.
(Example) When the original value is set to "10.000", "10.000 + 1.234 = 11.234" is displayed.

Note

(1) If the parameter "#11017 T-ofs set at run" (Tool compensation amount setting during automatic operation enabled) is set to "1", the workpiece coordinate system shift value can also be designated in the automatic operation mode or operation pause mode.

Operation method (Measure the workpiece coordinate system shift value of the Z axis (2nd axis) on the Monitor screen.)

(1) Select the basic tool used to measure the workpiece shift value with the manual numerical value command.
T1003
The T command can also be issued with MDI operation, etc. If one block contains multiple T commands, the last T command is used as the selected tool.

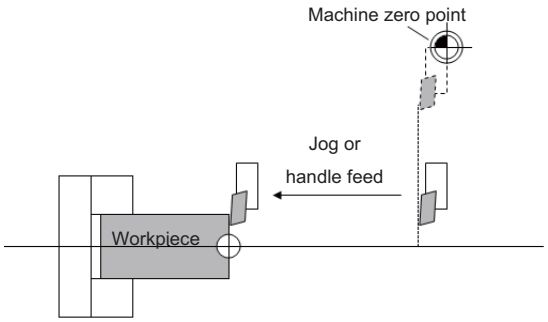
→ The manual numerical value command screen is displayed, and tool 1003 is selected.

(2) This displays the workpiece coordinate system shift.
<Monitor screen>
Press the menu [W-shift].
<Setup screen>
Press the menu [W-meas].

→ The workpiece coordinate system shift screen is displayed.

(3) Select the "Manual handle feed" mode or "Jog feed" mode.

(4) Select the Z axis, and manually move the tool to the work end face (program origin).



(5) Input the name of the axis to be measured, and the measurement value from the workpiece to the tool nose.
Z0.

→ When you input "Z0", the cursor moves to the Z axis column.
Value "Z0." is displayed in the input column.
If you input an axis name that does not exist, the cursor does not move.
The second and subsequent characters are not considered to be the axis name and, therefore, the cursor does not move.
(Example: If you enter "XZ", the cursor is moved on the X axis, but the cursor does not move on the Z axis.)

(6) Press the menu [Measure].

→ The measurement result is set to the workpiece coordinate system shift value pointed by the cursor.
The input column is cleared.

[Details on formula]

The measurement result is calculated in the following formula.

The workpiece base point offset is required to be set by MTB. The amount of workpiece base point offset is set to the parameter "#2701 Work base ofs".

When the parameter "#8709 Ext work sign rvs" is set to "0":

Measurement result = (Machine position of Z axis) - (Tool length compensation data of T-command tool) - (Input value) - (Workpiece base point offset)

When the parameter "#8709 Ext work sign rvs" is set to "1":

Measurement result = ((((Machine position of Z axis) - (Tool compensation data of T-command tool)) x (-1)) - (Input value)) - ((Workpiece base point offset) x (-1))

Select the tool compensation data used for the measurement with the parameter "#1226 aux10/bit0".

0: Tool length data + nose wear data
1: Tool length data

Select the tool No. to obtain the tool compensation data with the parameter "#1766 cfgPR16/bit3".

0: Tool compensation No. set on the tool management screen
1: Tool compensation No. set in the PLC device

Note

(1) If the parameter "#11017 T-ofs set at run" (Tool compensation amount setting during automatic operation enabled) is set to "1", the tool compensation amount data can be input in the automatic operation mode or operation pause mode.
(2) If you press the menu [Measure] while the axis name designated in the input section is different from that pointed at the cursor, the operation message "Setting error" is displayed.

6.9.2 Automatically Setting the Workpiece Coordinate System Shift Value

Carry out the automatic setting for the workpiece coordinate system shift value using the workpiece measurement signal.

Note

(1) Refer to the manual issued by MTB for details on each switch described in the operation procedures below. PLC devices for the signal of 1st part system are used for explanation.

Operation method

(1)	Carry out the reference position return. If the workpiece is measured without establishing the reference position return, the operation message "Meas axis not returned to ref. position" appears.	➡	Carry out the dog-type reference position return, and establish the coordinate system. In the absolute position detection specification, configure the initial setting in advance if the absolute position is not established.
(2)	Set the mode selection switch to the manual mode (handle or jog).		
(3)	Turn ON the tool measurement mode signal (YC21).	➡	"On mea" is displayed on the workpiece coordinate system shift screen of the setup screen.
(4)	Select whether to measure the workpiece coordinate system shift value on the main spindle side or sub spindle side. Use the tool presetter sub-side valid signal (YCDA) to select main side or sub side. <Note> ♦Do not change status of the switch until the measurement is completed.		
(5)	Issue the T command by MDI operation, etc., then select tool. Set the length data and wear data for the tool to be used beforehand.		(For MTB) Set the selected tool compensation No. in R register. The set R register differs depending on parameter setting and the state of the tool presetter sub-side valid signal.
(6)	Cut the workpiece face. This point is the program zero-point. <Note> ♦Do not move the tool in the Z axis direction after cutting the workpiece face.		
(7)	Display the workpiece coordinate system shift screen, and turn ON the workpiece coordinate measurement signal (Y9E1).	➡	The Z axis workpiece coordinate shift amount is automatically calculated by the machine value at the workpiece coordinate measurement signal is input and by the tool compensation data of the used tool. And the calculated value is set.
(8)	Turn OFF the tool measurement mode signal (YC21).	➡	The measurement of the workpiece coordinate system shift value is completed.

[Details on formula]

The workpiece coordinate system shift value is automatically calculated with the following formula.

The workpiece base point offset is required to be set by MTB. The amount of workpiece base point offset is set to the parameter "#2701 Work base ofs".

When the parameter "#8709 Ext work sign rvs" is set to "0":

Workpiece coordinate system shift amount = Machine coordinate value - Tool compensation data - Workpiece base point offset

When the parameter "#8709 Ext work sign rvs" is set to "1":

Workpiece coordinate system shift value = ((Machine coordinate value - Tool compensation data) x (-1)) – ((Workpiece base point offset x (-1))

Select the tool compensation data used for the measurement with the parameter "#1226 aux10/bit0".

0: Tool length data + nose wear data
1: Tool length data

[Compensation No. of selected tool]

The compensation number of the tool length and nose wear data used for automatic calculation is determined by the PLC ladder.

Precautions

(1) When the compensation No. is 0, the tool compensation amount will be calculated as "0".

(2) If the compensation No. exceeds the number of specified offset sets, the message "Offset No. not found" appears.

(3) If the input data exceeds the setting range, the message "Data range error" appears.

(4) If the data protection is valid, the message "Data protect" appears.

(5) Measurement is disabled for the part system with one-axis structure. The message "Can't measure" is displayed.

(6) When you perform the operations without displaying the workpiece coordinate system shift screen on the set up screen, no message is displayed.

6.10 Measuring the Workpiece Installation Error Data

Setup – 

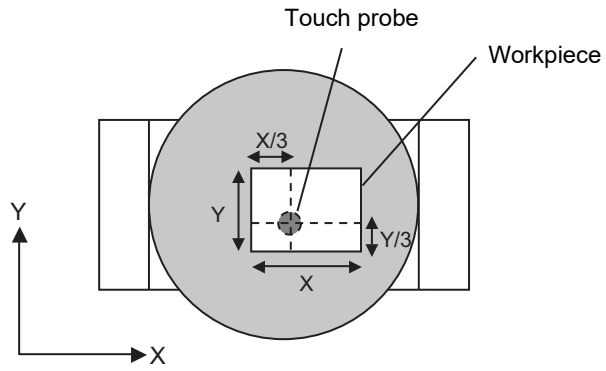
The workpiece installation error measurement screen is to measure the workpiece installation error and to set error offset amount. By setting the information of the workpiece to be measured or touch probe, etc. to be used for measurement in advance, the movement of the touch probe, reading the coordinate and calculation of the error are performed automatically. Also, you can set the calculated error to the offset amount of the specified workpiece installation coordinate system G54.4 Pn (n = 1 to 7).

This screen is displayed only when the optional specification of workpiece installation error compensation is valid in M system. This screen is a function for each part system.

Measurement procedure

(1) Place the workpiece and touch probe

Place the workpiece on the table and set each rotary axis to 0°. In this state, manually move the touch probe to one third of X width from the end surface of the workpiece on X negative side and one third of Y width from the end surface of the workpiece on Y negative side as in the figure below. Moreover, manually move the touch probe close to the workpiece just until it contacts the workpiece from Z axis positive direction.

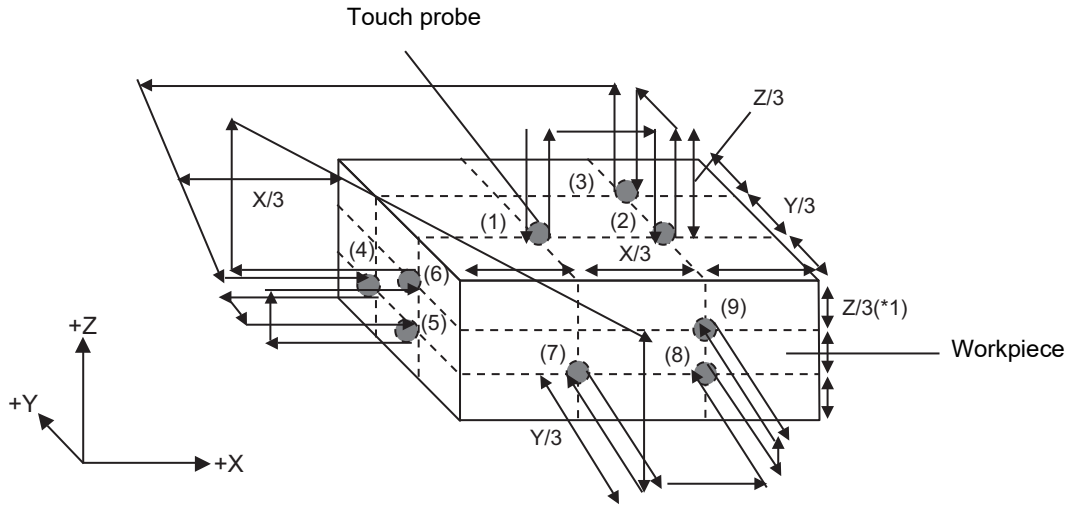


(2) Setting the measurement conditions

Press the menu [Measure cond.] on the workpiece installation error measurement screen to set the information of the touch probe or workpiece to be used for measurement and the common variable to store the measurement result.

(3) Perform the measurement

After pressing the menu [Start measure] to set it to the measurement state, perform cycle start. After the measurement is started, automatically measure three points each for (1) to (3) on the top face, (4) to (6) on the left side face and (7) to (9) on the front face as in the figure below. After the measurement is completed, the calculated installation error is displayed on the screen.



(4) Setting the measurement result

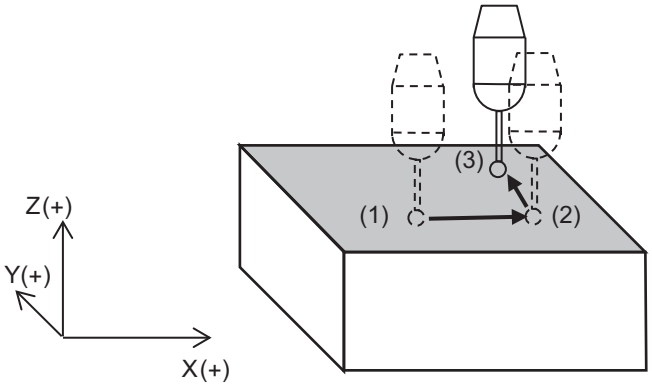
Press the menu [Write WK err.] to set the measurement result displayed on the screen to the offset amount of the workpiece installation coordinate system G54.4Pn set in the measurement condition.

Measurement operation descriptions

Press the cycle start to measure three points each on the top face, side face and front face (five faces when the workpiece zero point is the workpiece center) and the workpiece installation error is calculated.

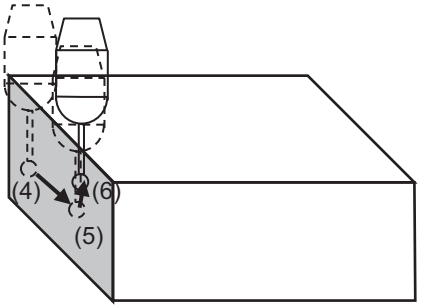
■Measure from the top face

Measure three points on the top face to calculate the inclination on the top face.



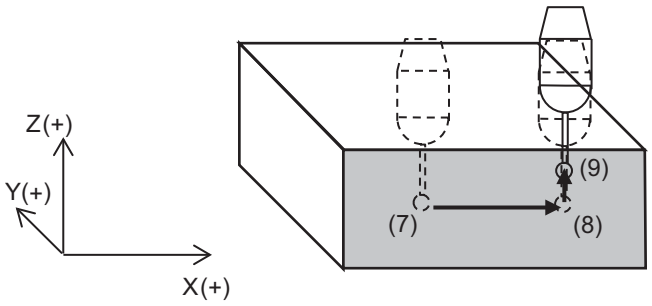
■Measure from the side face

Measure three points on the side face to calculate the inclination on the side face.

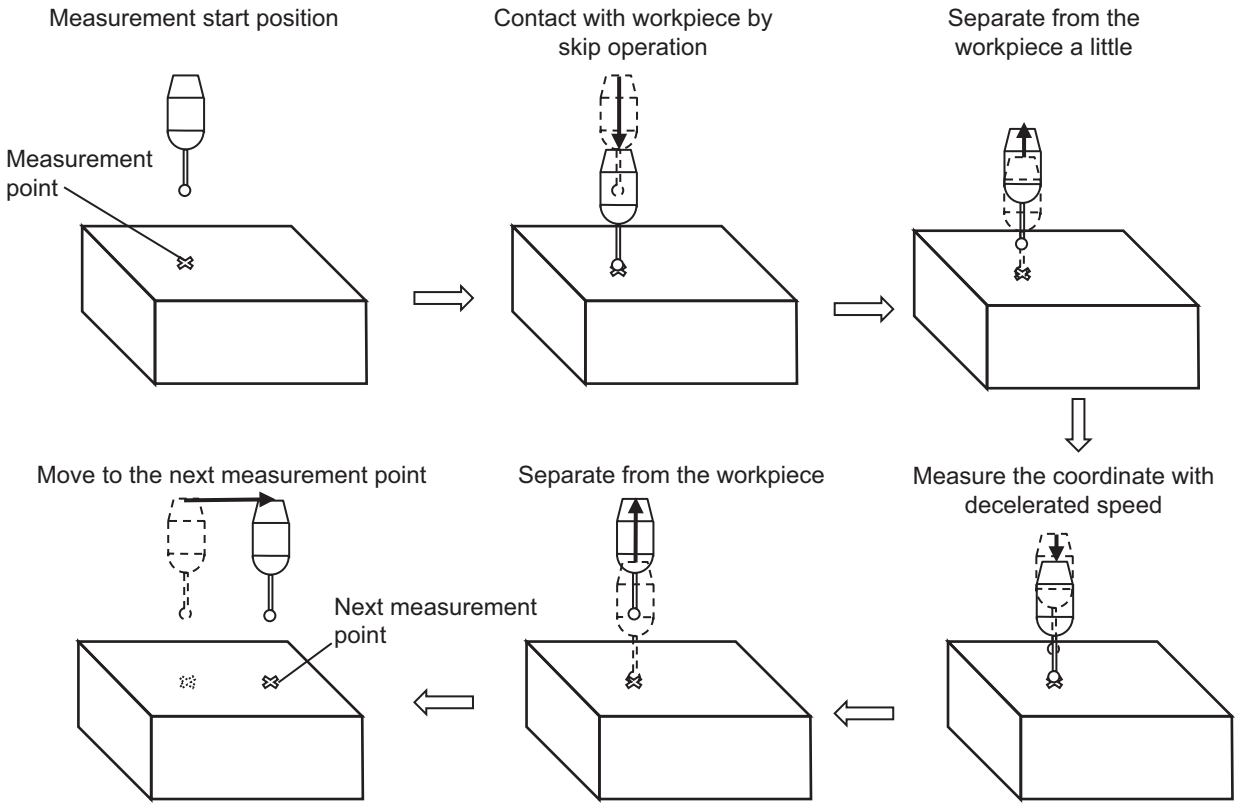


■Measure from the front face

Measure three points on the front face to calculate the inclination on the front face.



On each measurement point, after contacting the touch probe to the workpiece with skip operation, separate the touch probe from the workpiece once and then measure the coordinate on the workpiece with decelerated skip operation.

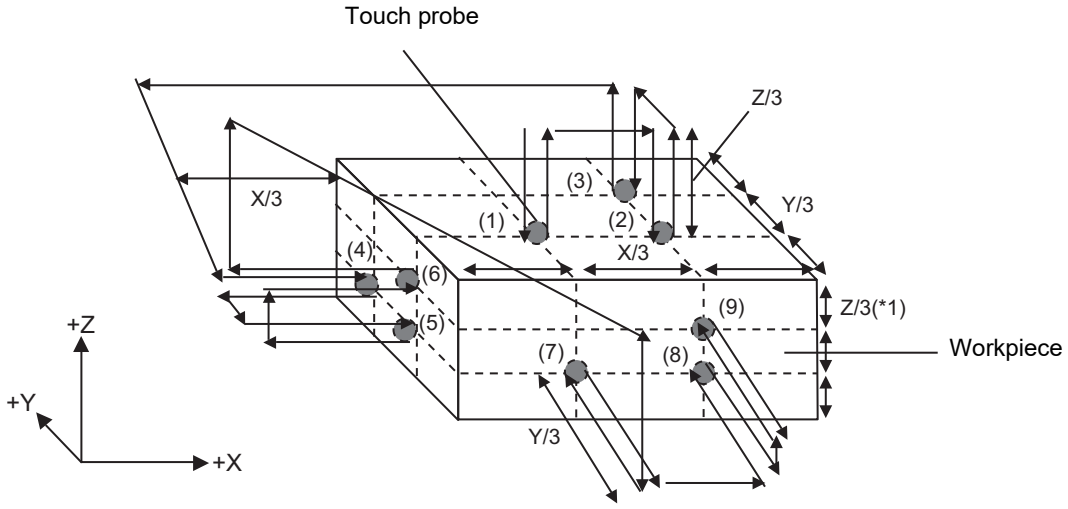


The measurement is started from the position one third away from each side in the direction perpendicular to the measurement face for each measurement point.

When the measurement face is switched, the touch probe moves at Z/3 height from the top face of workpiece.

If the workpiece is greatly tilted and the height error between the measurement points on the same face is Z/3 or more, the sensor may be damaged.

When the width in the X, Y, Z direction of the workpiece is X, Y, Z respectively, the sensor tip moves as in the following (1) to (9).

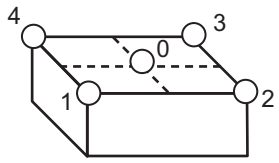


Note

(1) Assuming that the Z direction width of the workpiece is Z and the stylus length is S, when S is smaller than Z, the measurement height of side face and front face become S third and S * two third from the top face.

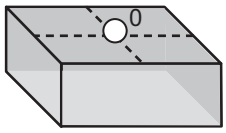
Depending on which point on the workpiece the workpiece zero point is set on, the face to be measured is switched between the left face/right face, or front face/back face.

Selectable workpiece zero point position (0 to 4)

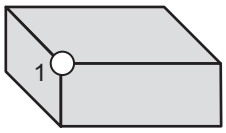


Switching measurement face at each workpiece zero point position

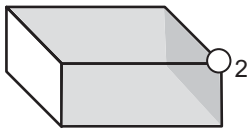
Workpiece zero point position: 0
Measurement face: top, left, back, right and front



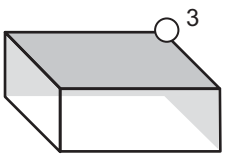
Workpiece zero point position: 1
Measurement face: top, left and front



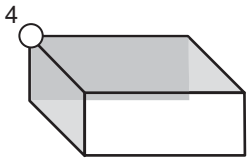
Workpiece zero point position: 2
Measurement face: top, right and front



Workpiece zero point position: 3
Measurement face: top, right and back

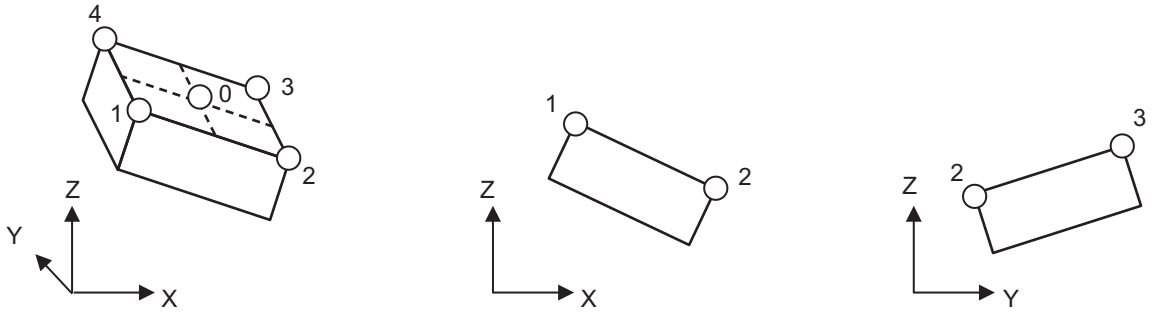


Workpiece zero point position: 4
Measurement face: top, left and back



When the workpiece is greatly tilted, confirm the measurement face in the figure on the previous page and set the workpiece zero point so than the stylus bar portion does not contact the workpiece.

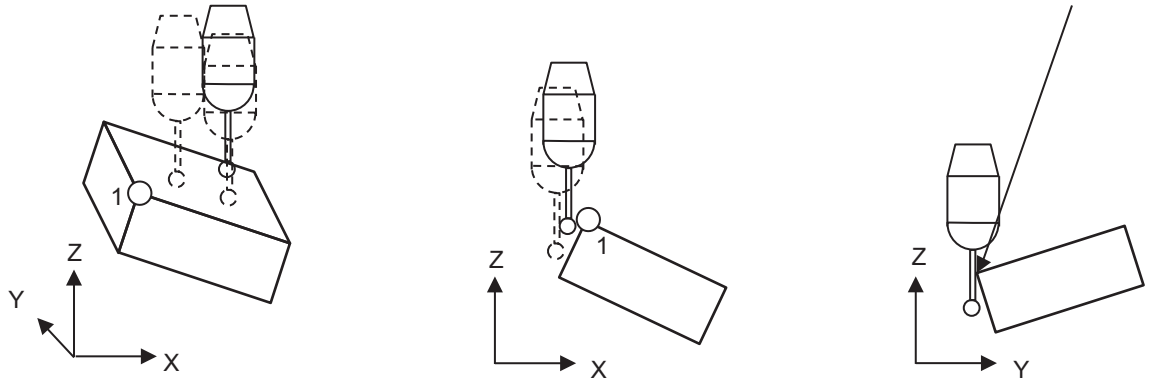
Tilted workpiece



Measure with Workpiece zero point position: 1

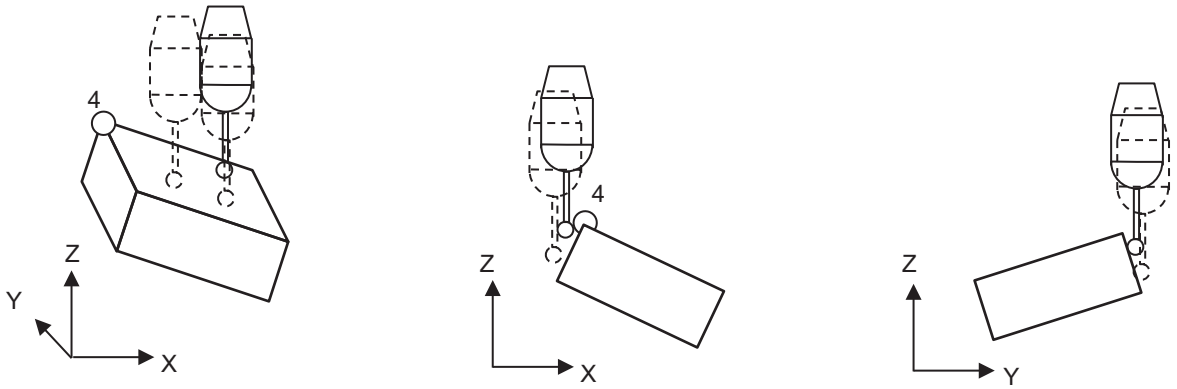
Top | Left | Front

The stylus bar portion contacts the workpiece angle before the stylus ball contacts the face.



Measure with Workpiece zero point position: 4

Top | Left | Back



Parameter setting examples

Example of parameter setting when measuring with the following configuration.

[Sensor information]

Stylus sphere diameter: 5 mm
Stylus length: 50 mm
Overtravel amount: 21 mm (in XY direction), 11 mm (in Z direction)
Tool length: 176 mm

[Workpiece information]

X width: 70 mm
Y width: 100 mm
Z width: 40 mm

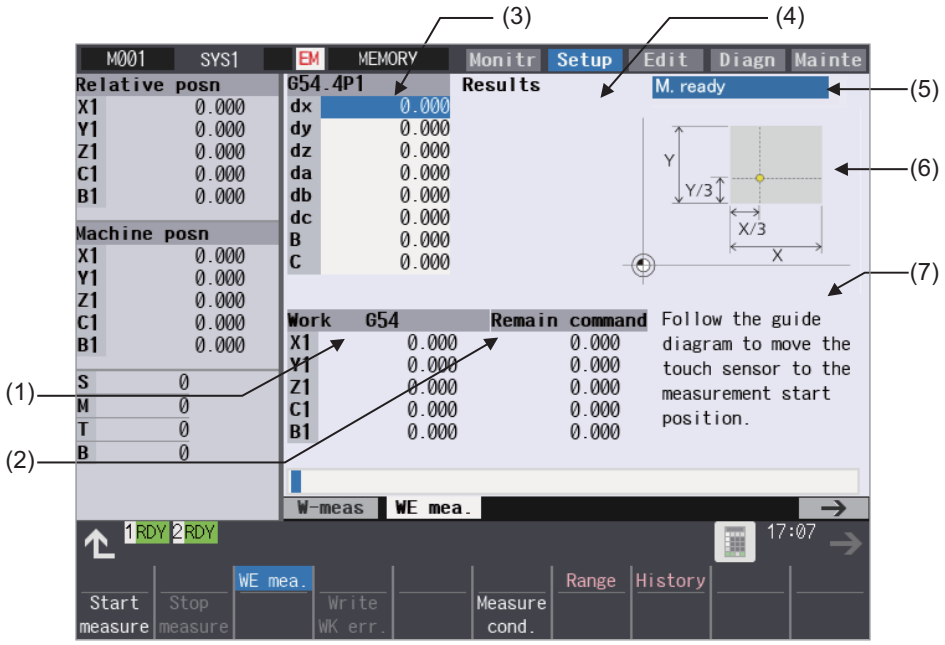
[NC setting information]

Tool length: Set the sensor tool length (176 mm) to tool No. 2
Common variable No. to be used: 800 to 807
Destination for writing error compensation amount: G54.4 P3
Workpiece coordinate: G54.1 P5
Workpiece zero point: Center of the workpiece

Parameter setting examples

Parameter No.	Details	Value
#11700	Starting common variable No. for storage of error measurement result	800
#8717	WE measure wk zero	0
#8718	WE meas. wk coord	54.1
#8719	WE meas. ext wk No	5
#8720	WE mea. wk X-width	70
#8721	WE mea. wk Y-width	100
#8722	WE mea. wk Z-width	40
#8723	WK inst. err comp#	3
#8724	Sensor T L comp No	2
#19101	Stylus ball dia.	5
#19102	Stylus length	50
#19103	Sensor overtravel	11

6.10.1 Screen Configuration



Display items

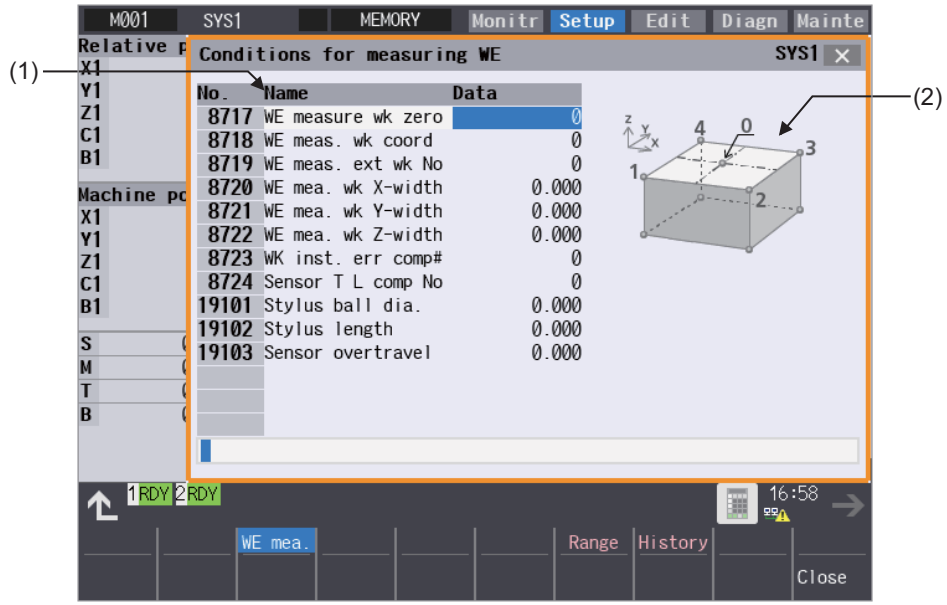
	Display items	Details
(1)	Workpiece coordinate G54 counter	This displays the workpiece coordinate G54 counter.
(2)	Remaining command counter	This displays the remaining command counter.
(3)	Workpiece installation error compensation amount	This displays the error offset amount of the workpiece installation coordinate system (G54.4Pn) corresponding to the set workpiece installation error No. Also, the error offset amount can be set. For datails, refer to "Display items" of "6.8.6 Setting the Offset Amount of Workpiece Installation Error".
(4)	Measurement result	This displays the measurement result of workpiece installation error. <Note> ♦This displays the value of eight sets of common variables from the head No. of common variable set in the measurement condition.
(5)	Measurement status	This displays the status of the followings according to the measurement status. ♦M. ready This shows the measurement can be executed. When the cycle start is performed in this state, the measurement is started. ♦On mea This shows the measurement is in progress. ♦Complete This shows the measurement is completed.
(6)	Guide graphic	This displays the installation position of touch probe when the measurement is started.
(7)	Guidance	This displays the guidance related to the measurement.

Menus

Menus	Details
Start measure	The measurement is ready. When the cycle start is performed in this state, the measurement is started. When the measurement is completed, the measurement result is updated.
Stop measure	This interrupts the measurement. When the measurement is interrupted, the measurement result is not updated.
Write WK err.	This sets the measurement result to the workpiece installation error offset amount of the specified No.
Measure cond.	This displays the pop-up window to set the measurement condition.

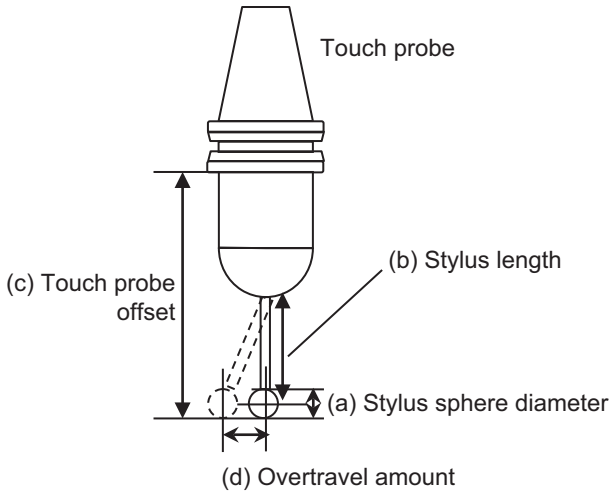
6.10.2 Setting the Measurement Condition

When the menu [Measure cond.] is selected, the pop-up window appears. Set the information of the workpiece and touch probe to be used for the measurement.



Display items

Display items		Details
(1)	Measurement condition parameters	This sets the parameters required for the measurement.
	WE measure wk zero	This sets the workpiece zero point No. Setting range: 0 to 4
	WE meas. wk coord	This sets the workpiece zero point setting position at the time of the workpiece installation error measurement. Setting range: 0 to 4
	WE meas. ext wk No	This sets No. after P of G54.1 when the workpiece coordinate of the workpiece where the workpiece installation error measurement is performed is extended workpiece coordinate system. Setting range: 0, 54 to 59
	WE mea. wk X-width	This sets the width in X direction of the workpiece. (*1) Setting range: 0.000 to 99999.999 (mm)
	WE mea. wk Y-width	This sets the width in Y direction of the workpiece. (*1) Setting range: 0.000 to 99999.999 (mm)
	WE mea. wk Z-width	This sets the width in Z direction of the workpiece. (*1) Setting range: 0.000 to 99999.999 (mm)
	WK inst. err comp#	This sets the workpiece installation error No. to write the measurement result. (Example) For G54.4P1, set "1". Setting range: 0 to 7
	Sensor T L comp No	This sets the offset No. of the touch probe to be used for the measurement. (Figure (c) below) Setting range: 0 to 999
	Stylus sphere dia.	This sets the diameter of stylus sphere of the touch probe. (Figure (a) below) Setting range: 0.000, 1.000 to 10.000 (mm)
	Stylus length	This sets the length of stylus of the touch probe. (Figure (b) below) Setting range: 0.000 to 99999.999 (mm)
	Sensor overtravel	This sets the overtravel amount (overtravel allowance amount and over stroke) of the sensor to be used for the measurement. (Figure (d) below) Setting range: 0.000 to 100.000 (mm)
(2)	Guide graphic	This displays the guide drawing of workpiece zero point position.



(*1) If the range where the touch probe can be contacted is small due to the jog, etc., specify the width of the part outside the jig.

Operation method

(1) Press the menu [Measure cond.].

(2) Set the workpiece installation error No.
With the cursor at the workpiece installation error No., input the numerical value and press [INPUT] key.
(Example) 3 [INPUT]
→ The numerical value input to the workpiece installation error No. is set. The cursor moves to the next line.

(3) In the same way, set each measurement condition. After the setting is completed, select the menu [Close] to close the pop-up window.

6.10.3 Performing Workpiece Installation Error Measurement

Operation method (Performing the measurement)

(1) According to the guide graphic, move the touch probe manually to the measurement start position.

(2) Press the menu [Start measure].
→ "M. Ready" is displayed in the measurement state. "EM" is displayed in the NC status.

(3) Press the cycle start button.
→ The measurement status becomes "On mea" and the measurement is performed automatically.
When the measurement is completed, the measurement result is updated. Also, "Complete" is displayed in the measurement status and the NC status "EM" disappears.

(4) Press the menu [Write WK err.].
→ The confirmation message "OK to write measurement results?(Y/N)" appears.

(5) Press the [Y] or [INPUT] key.
→ The measurement result is set to the offset amount of the workpiece installation coordinate system G54.4Pn set in the measurement condition.

Note

(1) Confirm the skip signal (X6F8) is input when the touch probe is contacted before the measurement is performed. If the measurement is performed in the state that the skip signal cannot be input, the touch probe may be damaged.

(2) If reset or emergency stop is performed during the measurement, the measurement is interrupted. In this case, the measurement result is not updated.

(3) If the menu [Start measure] is pressed during the automatic operation, the operation message "Executing automatic operation" is displayed and the measurement is not started.

(4) If the menu [Write WK err.] is pressed during the automatic operation, the operation message "Executing automatic operation" is displayed and the offset amount cannot be set.

(5) If the offset amount of the workpiece coordinate system G54.4Pn (n=1 to 7) is set during the measurement, the operation message "Executing automatic operation" is displayed and the offset amount cannot be set.

(6) When the workpiece installation error measurement is performed in multiple part systems continuously, write the error after error measurement before starting the measurement in the next part system. If the next measurement is started without writing the error, the last measurement result is lost.

Operation method (Canceling the measurement)

(1) Press the menu [Stop measure]. → The menu [Stop measure] is highlighted and the confirmation message "Is it OK to discontinue measure?(Y/N)" is displayed.

(2) Press the [Y] or [INPUT] key. → The measurement preparation state is canceled. Also the highlight of the menu [Stop measure] is canceled and the NC status "EM" and the measurement status display "On mea" disappear.

6.10.4 Precautions for Workpiece Installation Error Measurement

Availability of check mode with other functions

This section explains regarding the operation when the workpiece installation error measurement is performed with other functions.
Depending on the combination, the sensor may be damaged or the error may not be calculated correctly.

■Function where the coordinate system changes

When the function where the coordinate system changes is valid, the workpiece installation error measurement cannot be performed correctly. If performed, the sensor may be damaged or the error may not be calculated correctly.

Do not perform the workpiece installation error measurement combined with the following functions.

- Polar coordinate interpolation (G12.1)
- External workpiece coordinate offset (The setting value is not "0".)
- Local coordinate system (G52)
- Origin set
- Workpiece position offset for rotary axis (G54.2)
- Workpiece coordinate system shift
- Coordinate rotation by program (G68)
- Coordinate rotation by parameter
- 3-dimensional coordinate conversion (G68)
- Polar coordinate command (G16)
- Tool center point control (G43.4, 43.5)
- Inclined surface machining command (G68.2)
- R-navi (during the surface selection, during indexing)

■Function where the path changes

When the function where the path changes is valid, the workpiece installation error measurement cannot be performed correctly. If performed, the sensor may be damaged or the error may not be calculated correctly.

Do not perform the workpiece installation error measurement combined with the following functions.

- Tool length compensation (G43, 44)
- Tool position compensation (G43.7)
- Tool radius compensation (G38, 39, 41, 42)
- 3-dimensional tool radius compensation (G41.2, 42.2)
- Nose R (G41, 42, 46)
- Scaling (G51)
- The parameter setting mirror image
- Mirror image by external input
- Mirror image (G50.1)
- Normal line control (G41.1, 42.1)
- Workpiece installation error compensation (G54.4)

■Function where the axes switch

When the function where the axes switch is valid, the workpiece installation error measurement cannot be performed correctly. If performed, the sensor may be damaged or the error may not be calculated correctly.

Do not perform the workpiece installation error measurement combined with the following functions.

- ♦Axis name change
- ♦Mixed control (cross axis control)
- ♦Arbitrary axis exchange control
- ♦Control axis superimposition
- ♦Arbitrary axis superimposition control

■Buffer correction

Buffer correction cannot be executed while the workpiece installation error measurement is performed.

■Restart search

Restart search cannot be executed for the workpiece installation error measurement. If executed, the restart search is executed for the program before the workpiece installation error measurement is performed.

■Collation and stop

Collation and stop cannot be executed for the workpiece installation error measurement.

■Chopping

If the workpiece installation error measurement is performed during chopping, the sensor may be damaged. Error is not calculated correctly.

■3D machine interference check

If the tool length of the sensor is not set to the tool length of 3D interference check, the sensor may be damaged or the measurement may be stopped.

Precautions

(1) Confirm the skip signal is input with the touch probe before the workpiece installation error measurement is performed. If the workpiece installation error measurement is performed in the state that the skip signal cannot be input, the sensor may be damaged.

(2) When "0006 H/W stroke end axis exists" occurs, change the setting position of the workpiece.

(3) When "0007 S/W stroke end axis exists" occurs, correct the parameter soft limit I (#2013 and #2014). Also, change the setting position of the workpiece.
If the workpiece installation error measurement is performed in the following state, the measurement is performed in the state that the compensation is valid, the sensor may be damaged and the measurement result may not be correct.

- ♦When the coordinate system is shifted due to manual interruption with manual absolute OFF
- ♦When the compensation by the coordinate system shift such as local coordinate system setting, workpiece coordinate system shift and external workpiece coordinate offset, various coordinate conversions or various offsets is valid

(4) If the workpiece installation error measurement is performed in the state the parameter or PLC signal which applies the compensation to the coordinate system, such as "mirror image by parameter setting", "mirror image by external input" or "coordinate system rotation by parameter", is set, the measurement is performed with the compensation valid. This may cause a sensor damage and the measurement result may not be correct.

(5) Single block stop cannot be performed during the workpiece installation error measurement.

(6) Perform the measurement in the state that the measured machine error compensation amount is set to the compensation data and the machine error compensation is valid.

(7) Measurement related parameters cannot be changed during cycle start. When changing the measurement related parameters, cancel the measurement before changing them.

(8) When changing the mode to the one other than memory mode in the state "On mea", the operation error "M01 Invalid op mode in err measure 0246" occurs and the manual interruption, automatic operation handle interruption, MDI interruption, etc. cannot be performed. After changing the mode to the memory mode, press the cycle start button to restart the error measurement.

(9) When the measurement macro is initialized or the measurement is started, do not change the parameter "#11701 WEM Mcr No" (Macro No. for workpiece installation error measurement). If changed, the sensor may be damaged.

(10) After the error measurement is completed, if the next measurement is started without writing the error, press the menu [Stop measure] to stop the measurement and then press the menu [Start measure].

(11) The workpiece installation error measurement can be performed when the axis names are X, Y and Z only. With other axis configuration, the measurement cannot be performed.

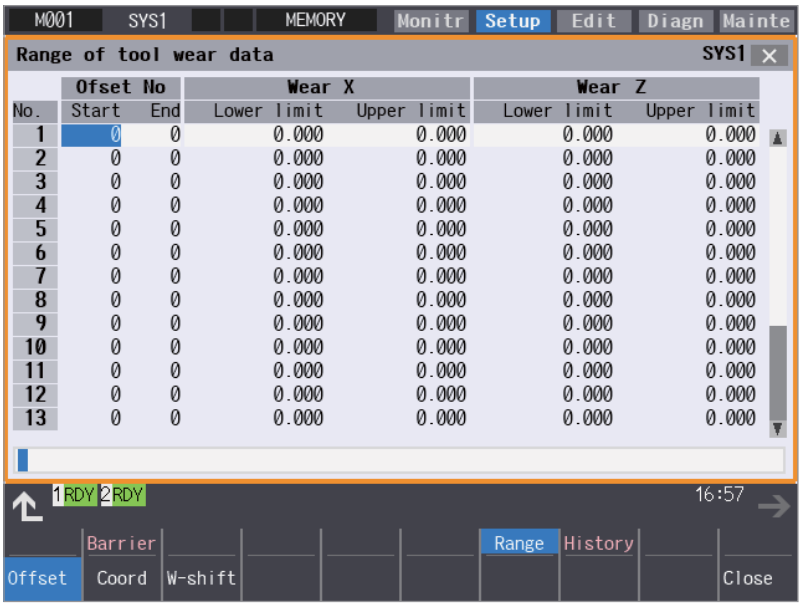
(12) This function does not support multiple part systems.

6.11 Designating the Input Range (Range Setting Screen)

Setup – Set range (For 15-type/19-type display unit)

Setup – Range (For 8.4-type/10.4-type display unit)

Available range of the offset data for tools and the workpiece coordinate system varies depending on the size or structure of the machine you are using.
This function enables you to customize the input range to be matched your machine. It is useful to eliminate input errors of offset data.
You can set available input range (lower limit and upper limit), which is used for each tool compensation screen, workpiece coordinate system offset screen and workpiece coordinate system shift screen, on the screen below.



Menus

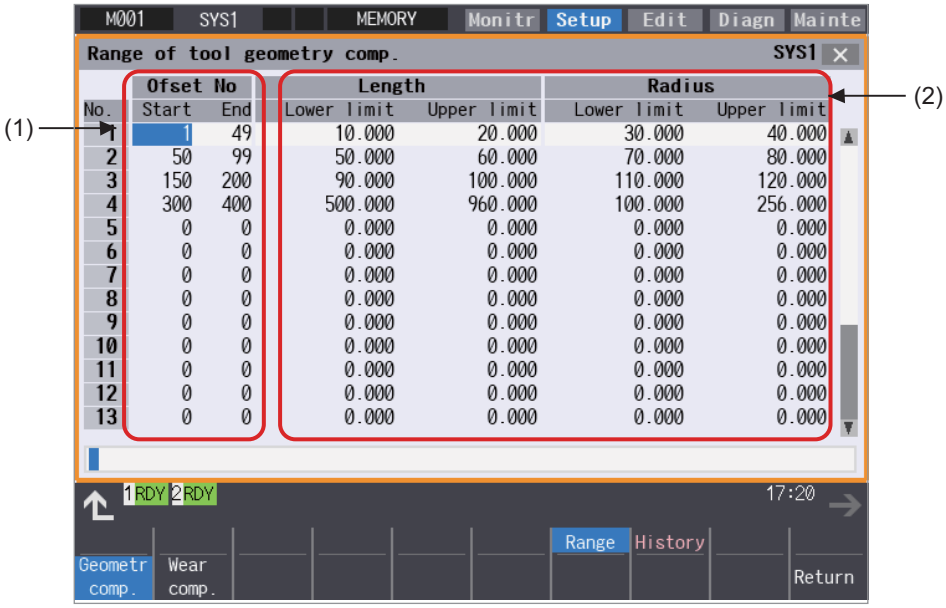
Menus	Details
Offset	Displays the range setting screen for the tool compensation data.
Coord	Displays the range setting screen for the workpiece coordinate system offset amount.
W-shift	Displays the range setting screen for the workpiece coordinate system shift amount. <Note> ♦This menu is displayed when the parameter "#11056 Workshift invalid" is set to "0" for L system.
Close	This closes the pop-up window and quits this function.

6.11.1 Designating the Input Range of the Tool Compensation Amount

You can set the input range (lower limit and upper limit) of the tool compensation data on the screen displayed by pressing [Range] - [Offset].
The input range check of the tool Nos. which are designated for "Start" and "End" is executed for the set values of lower limit and upper limit.
Maximum 20 ranges (lower limit and upper limit) can be set.
The screen configuration to be displayed varies depending on machining system (M system or L system) or MTB specifications.
Here explains using examples of the tool compensation type II (M system) and the tool compensation type III (L system).

■Tool compensation type II (M system)

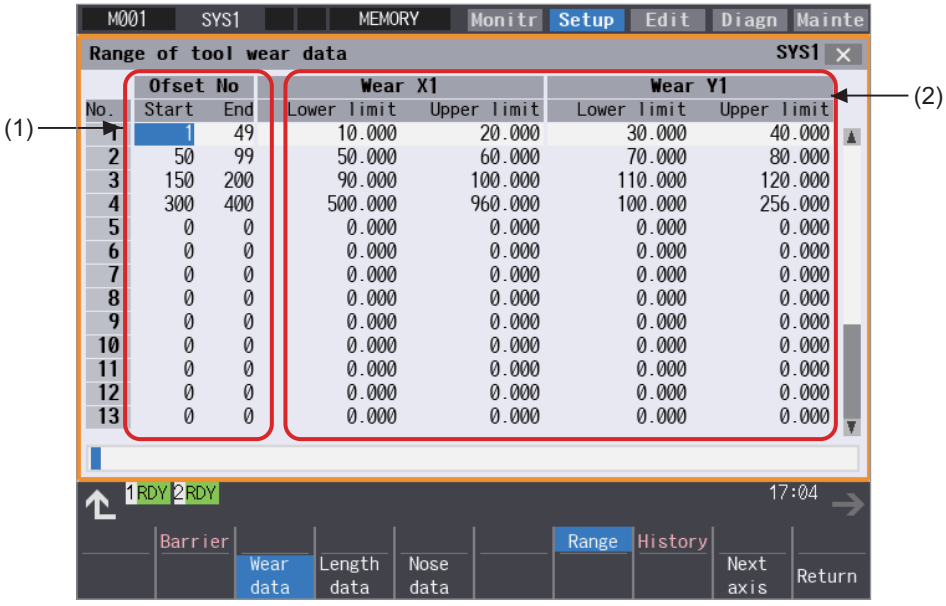
You can switch to display the shape compensation amount (tool length and radius dimension) or the wear compensation amount (length wear and radius wear) by pressing sub menus with this compensation type.



No.	Ofset No Start	Ofset No End	Length Lower limit	Length Upper limit	Radius Lower limit	Radius Upper limit
1	1	49	10.000	20.000	30.000	40.000
2	50	99	50.000	60.000	70.000	80.000
3	150	200	90.000	100.000	110.000	120.000
4	300	400	500.000	960.000	100.000	256.000
5	0	0	0.000	0.000	0.000	0.000
6	0	0	0.000	0.000	0.000	0.000
7	0	0	0.000	0.000	0.000	0.000
8	0	0	0.000	0.000	0.000	0.000
9	0	0	0.000	0.000	0.000	0.000
10	0	0	0.000	0.000	0.000	0.000
11	0	0	0.000	0.000	0.000	0.000
12	0	0	0.000	0.000	0.000	0.000
13	0	0	0.000	0.000	0.000	0.000

■Tool compensation type III (L system)

You can switch to display the wear data, tool length data or nose data by pressing sub menus with this compensation type.



No.	Ofset No Start	Ofset No End	Wear X1 Lower limit	Wear X1 Upper limit	Wear Y1 Lower limit	Wear Y1 Upper limit
1	1	49	10.000	20.000	30.000	40.000
2	50	99	50.000	60.000	70.000	80.000
3	150	200	90.000	100.000	110.000	120.000
4	300	400	500.000	960.000	100.000	256.000
5	0	0	0.000	0.000	0.000	0.000
6	0	0	0.000	0.000	0.000	0.000
7	0	0	0.000	0.000	0.000	0.000
8	0	0	0.000	0.000	0.000	0.000
9	0	0	0.000	0.000	0.000	0.000
10	0	0	0.000	0.000	0.000	0.000
11	0	0	0.000	0.000	0.000	0.000
12	0	0	0.000	0.000	0.000	0.000
13	0	0	0.000	0.000	0.000	0.000

Display items

Display items		Details
(1)	Compensation No. range designation	Designate the range of tool No. which is applied to the range between lower limit and upper limit set in the item (2). Setting range: 0 to 999
(2)	Compensation data range designation	Designate the range of the compensation data between lower limit and upper limit. The display items changes depending on the tool compensation type. For details of each setting item, refer to "6.4 Setting Tool Compensation Amount (Tool Offset Screen)".

Note

(1) The setting is disabled in the following cases, and the range check is executed with default values.

- "0" is set to either "Start" or "End".
- The value set to "Start" is greater than that of "End".
- Tool No. designated in "Start" or "End" also exists in the other range.
- The value set to "Lower limit" is greater than that of "Upper limit".
- "0" is set to both "Start" and "End".

Menus

Tool compensation type I

Menus	Details
Return	This returns to the [Range] menu.

Tool compensation type II

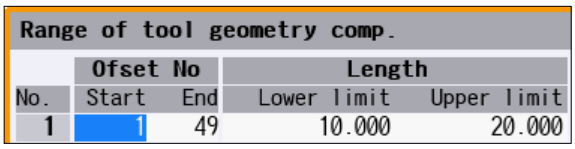
Menus	Details
Geometr comp.	This switches the display to the shape compensation amount (tool length and radius dimension).
Wear comp.	This switches the display to the wear compensation amount (length wear and radius wear).
Return	This returns to the [Range] menu.

Tool compensation type III

Menus	Details
Wear data	This switches the display to the wear data.
Length data	This switches the display to the tool length data.
Nose data	This switches the display to the nose data.
Next axis	This displays the compensation data of the next axis.
Return	This returns to the [Range] menu.

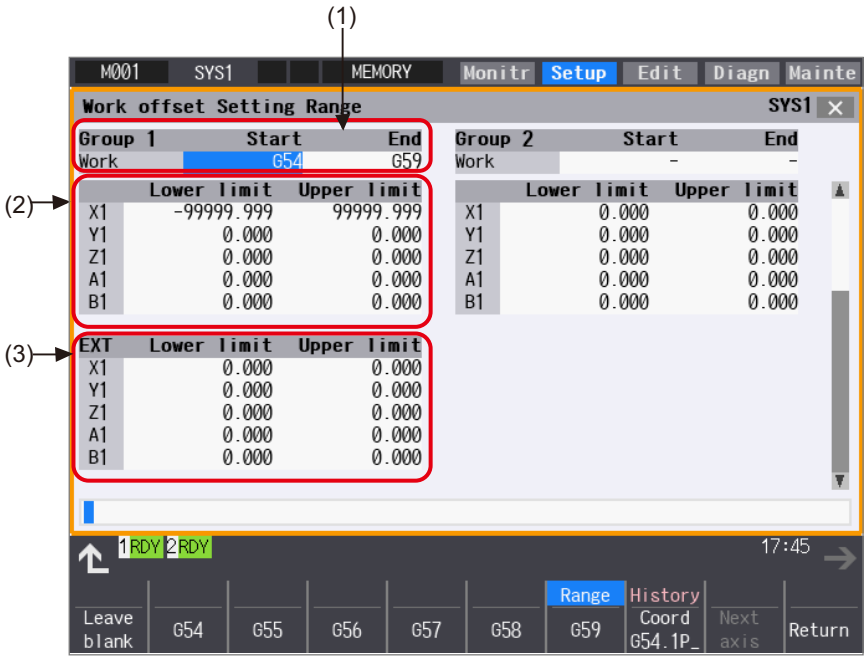
Setting example

In the setting example below, the input range check of the tool length between "10.000" and "20.000" is executed for the tool No. 1 to 49.



6.11.2 Designating the Input Range of the Workpiece Coordinate System Offset Amount

You can set the input range of the workpiece coordinate system offset data on the screen displayed by pressing [Range] - [Offset].
"Group 1" to "Group 6" are ready to set, and you can set the input range (lower limit and upper limit) for each group.
The input range (lower limit and upper limit) of the offset data for the external workpiece coordinate system can be set to "EXT".



Display items

	Display items	Details	Setting range
(1)	Coordinate system range designation	Designate the range of the coordinate system selecting from the menu G54 to G59, G54.1P which is applied to the range (lower limit and upper limit) set in the item (2). When you want to designate a single coordinate system, input the same coordinate system to both "Start" and "End". (Example 1) When you designate the range of workpiece coordinate system G54 to G57, input "G54" to "Start" and "G57" to "End". (Example 2) When you designate only the coordinate system G58, input "G58" to both "Start" and "End".	- (None) G54 to G59 G54.1P1 to G54.1P300
(2)	Coordinate system offset data range designation	Set lower limit and upper limit of the workpiece coordinate system offset amount.	-99999.999 to 99999.999 (mm) (*1)
(3)	External coordinate system offset data range designation	Set lower limit and upper limit of the external workpiece coordinate system offset amount	-99999.999 to 99999.999 (mm) (*1)

(*1) The setting range here is the value when the minimum command unit is 1μm (#1003 = B, #1041 = 0).

Note

(1) The setting is disabled in the following cases, and the range check is executed with default values.

- "-" is set to either "Start" or "End".
- The coordinate system set to "Start" is greater than that of "End".
- Any of the workpiece coordinate system (G54 to G59) and the extended workpiece coordinate system (G54.1) are mixed to set to "Start" and "End".
- The coordinate system designated in "Start" or "End" also exists in the other group.
- The value set to "Lower limit" is greater than that of "Upper limit".
- "0" is set to both "Start" and "End".

Menus (When the cursor is at the setting field of the offset data range for coordinate system)

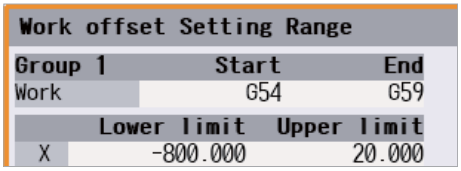
Menus	Details
Leave blank	This sets "-" to "Start" or "End". None of the workpiece coordinate system is set.
G54	This sets any of the workpiece coordinate system "G54" to "G59" to "Start" or "End".
G55	
G56	
G57	
G58	
G59	
Coord G54.1 P_	This sets the extended workpiece coordinate system "G54.1 Pn" to "Start" or "End". Input a value to P number by using numeric key.
Next axis	This displays the data of the next axis. This switches displayed axes to the next 5 axes. <Note> ♦If the number of axes is 5 or less, this menu is grayed out and non-selectable.
Return	This returns to the [Range] menu.

Menus (When the cursor is at the setting field of the offset data range for coordinate system)

Menus	Details
Next axis	This displays the data of the next axis. This switches displayed axes to the next 5 axes. <Note> ♦If the number of axes is 5 or less, this menu is grayed out and non-selectable.
Return	This returns to the [Range] menu.

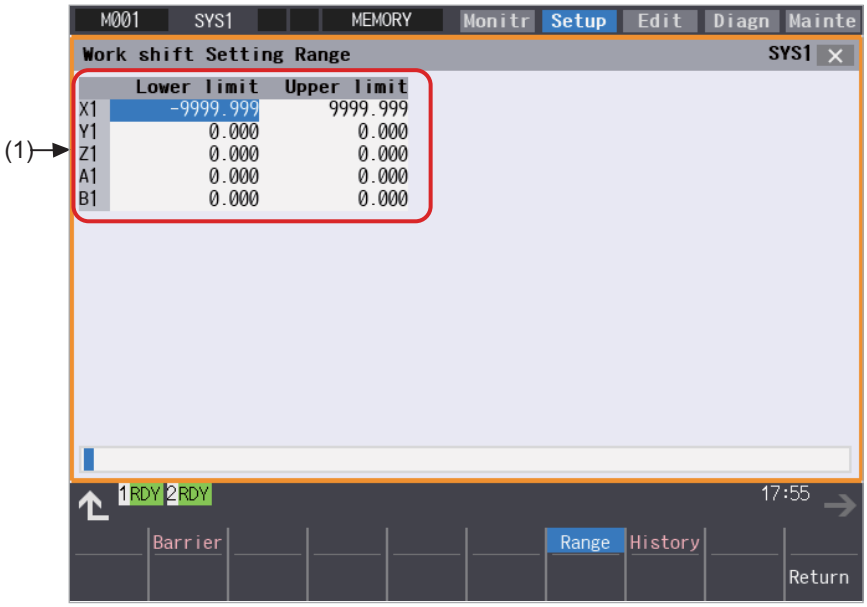
Setting example

In the setting example below, the input range check of the offset data for X axis between "-800.000" and "20.000" is executed for the workpiece coordinate system "G54" to "G59".



6.11.3 Designating the Input Range of the Workpiece Coordinate System Shift Amount

You can set the input range of the workpiece coordinate system shift data (lower limit and upper limit) on the screen displayed by pressing [Range] - [W-shift].



Display items

Display items		Details	Setting range
(1)	Coordinate system shift data range designation	Set lower limit and upper limit of the workpiece coordinate system shift amount.	-99999.999 to 99999.999 (mm) (*1)

(*1) The setting range here is the value when the minimum command unit is 1µm (#1003 = B, #1041 = 0).

Note

(1) The setting is disabled in the following cases, and the range check is executed with default values.

- The value set to "Lower limit" is greater than that of "Upper limit".
- "0" is set to both "Start" and "End".

Menus

Menus	Details
Return	This returns to the [Range] menu.

7

Setting Parameters

The parameters contain the user and machine parameters.

This chapter explains the details and operations for the user parameters on the Setup screen.

Refer to "Alarm/Parameter Manual" for details of each parameter.

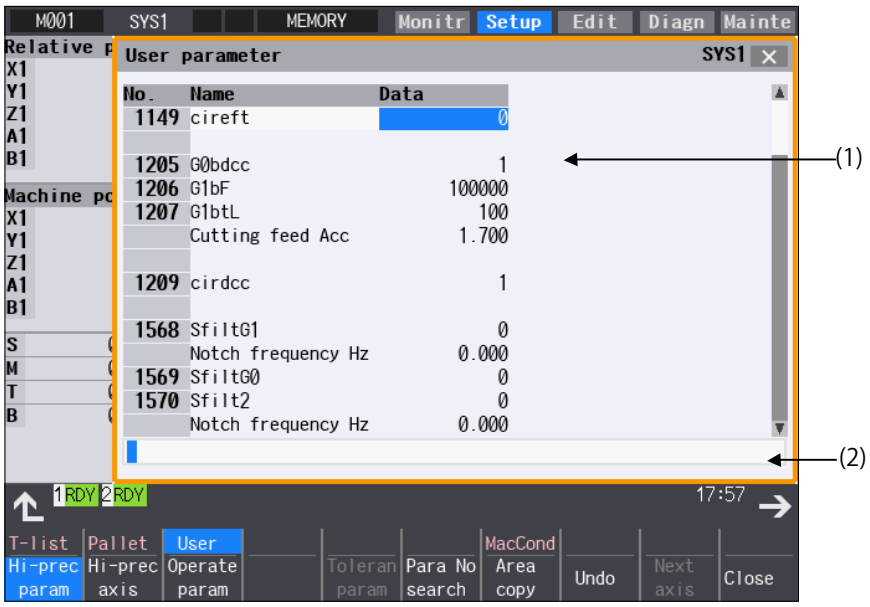
Parameters can be set on the maintenance (Mainte) screen.

7.1 Setting User Parameters

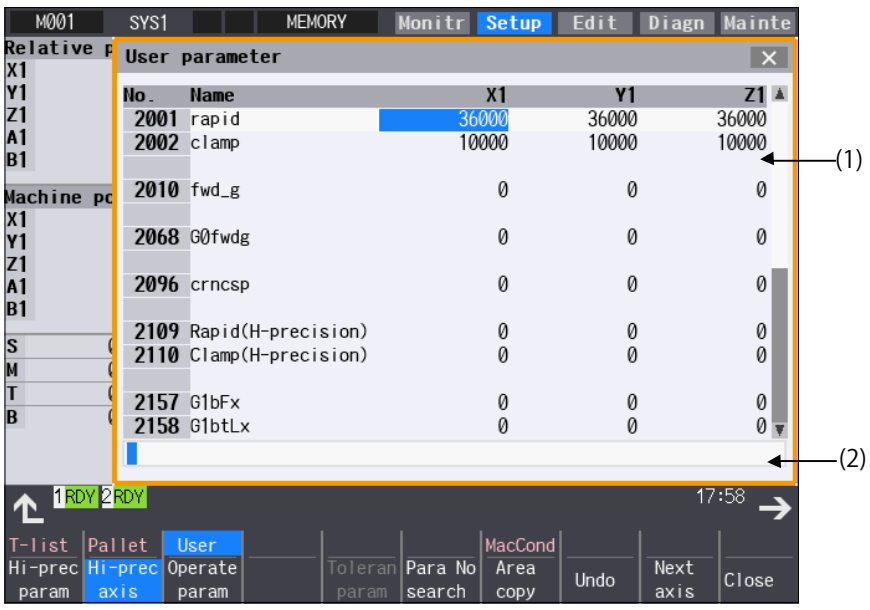
Setup – User

Press the menu [User] on the Setup screen to display various user parameters.

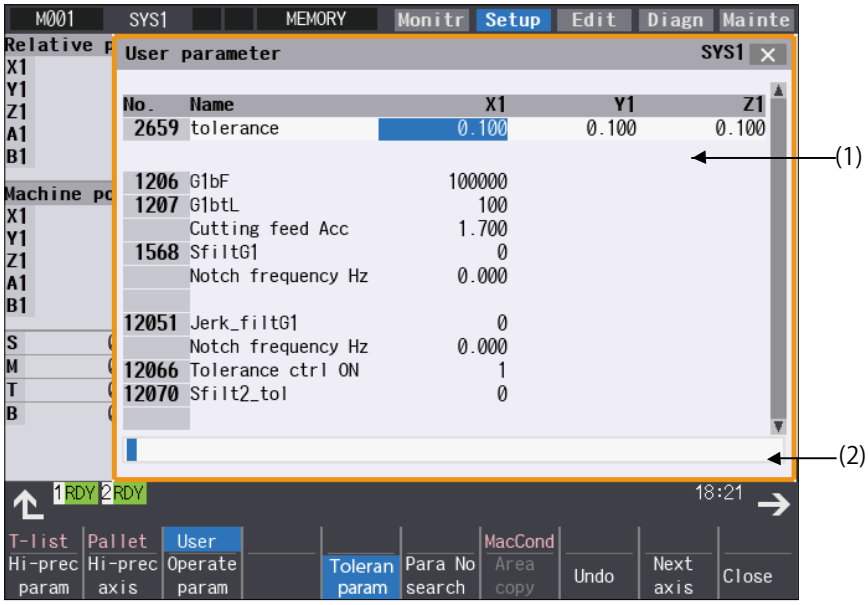
Display example by part system (Switch the part system using the $\leftrightarrow$ key.)



Display example by axis (Switch the fourth axis and after on the menu [Next axis].)



Display example by part system/axis



Display items

Display items		Details
(1)	Parameter display area	Press the parameter selection menu [Process param] or [Hi-prec param] etc. to change the parameter display contents. The screen layout can be switched by part system (with [$] key) or by axis.
(2)	Input section	The set key displays. Press the [INPUT] key to set the display data in the parameter where the cursor is located.

Menus

Menus	Details
Process param Fix Cycle Ctrl param 1 Ctrl param 2 I/O param Ethrnet param Link param Subpro stor Axis param Barrier data Hi-prec param Hi-prec axis Operate param Menu select Toleran param WLAN param	This changes the display of each parameter type. <Note> • The barrier data appears only for L system. •[Menu select] is displayed only when the parameter "#11032 Menu sel para lkof " (Menu select param setting enabled) is set to "1" or "2". •[WLAN param] is displayed only for M800VS/M80V Series.
Para No search	This selects an arbitrary parameter number. Set the parameter number and press the [INPUT] key to move that number to the top and display the parameter. The cursor also moves to that number.
Area copy	This copies the parameter setting values in the designated range. The range is designated with numbers.When the range is not designated, this copies the value of the parameter pointed by the cursor. After copying, the menu will be changed to [Area paste].

Menus	Details
Area paste	This pastes the range of parameters designated by [Area copy] to the corresponding parameters of the axis or part system pointed by the cursor. After pasting, the confirmation massage "Change the menu back to Area copy?(Y/N)" appears. Press the [Y] or [INPUT] key to go back to [Area copy]. When pressing the any key other than [Y] or [INPUT], the paste-enabled state will remain and you can continue pasting by pressing the menu [Area paste]. <Note> • While the confirmation message "Paste?(Y/N)" is displayed, the menu will be highlighted. •To go back to [Area copy] without pasting, switch the parameter type once.
Undo	This returns the last rewritten data to its previous value. <Note> • When the parameter "#8939 Undo confirm msg" is set to "1", the menu is highlighted while the confirmation message is displayed.
Next axis	This can be selected if the total number of part system display axes is 4 or more. This is used at the parameter screen with the array structure for each axis.
Close	This closes the pop-up window and quits this function.

7.1.1 Selecting the Parameter Number

There are two switching methods to display a screen by designating the parameter No. Use the parameter "#8975 No. search process" to switch the operation

Operation method ("#8975 No. search process" is "0") (Select "#1041 Initial inch")

(1) Press the menu [Para No search].

(2) Enter a parameter No. to be displayed, and then press the [INPUT] key.
1041 [INPUT]
➡ "#1041 Initial inch" is displayed at the head. The cursor moves to "#1041 Initial inch".

Operation method ("#8975 No. search process" is "1") (Select "#1041 Initial inch")

(1) Enter a parameter No. to be displayed.
1041

(2) Press the menu [Para No search].
➡ "#1041 Initial inch" is displayed at the head. The cursor moves to "#1041 Initial inch".

Note

(1) When the parameter "#8975 No. search process" is set to "1" and the cursor is at the parameter to be echoed back, the set value is displayed in the input column. Delete the set value in the input column, and then perform the menu [Para No search] operation.

7.1.2 Setting the Parameters

The method of setting the parameters is explained. Refer to "Alarm/Parameter Manual" for details of each parameter setting range.

Operation method (Normal) (Set "100" for Y1 axis of "#2001 rapid")

(1) Press the menu [Hi-prec axis].

(2) Use the [↑], [↓], [|←] and [→|] keys to move the cursor to the setting position.

(3) Input the value.
100 [INPUT]

Operation method (Batch data setting) (Set "4000" for Y1 axis and "3000" for Z1 axis for "#2002 clamp")

(1) Press the menu [Hi-prec axis].

(2) Use the [↑], [↓] keys to move the cursor to the setting position.

(3) Input the value.
Input format: 1st column/2nd column/3rd column
/4000/3000 [INPUT]

Note

(1) When a parameter is changed and [PR] displays at the top of the screen, reboot the machine to enable that parameter change.

(2) When the [INPUT] key is pressed without inputting a value, the parameter setting value is not changed, and the cursor moves.

(3) Parameters that can be set simultaneously must display within the currently displayed three columns.

(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.

(5) When the data protection is valid, the value cannot be set (including paste operation) for the high-accuracy control parameters (part system based), high-accuracy control axis parameters and tolerance parameters.

7.1.3 Copying and 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 [Area copy].

(2) Move the cursor to the position of the parameter to be copied and press the [INPUT] key. → Copying position is displayed in the selected area color. (If you move the cursor, the highlighted position can be confirmed.)
The menu highlight returns to normal and the menu will be changed to [Area paste].

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 [Area copy].

(3) Designate the copy range.
Format: First No./Last No.
8701/8705 [INPUT] → The range to be copied is displayed in the selected area color.
The menu highlight returns to normal and the menu will be changed to [Area paste].

An "E" can be used if the last number to be copied is the last number in the currently displayed parameter type.
(Example) 8701/E

Note

(1) The copy range remains enabled immediately before you change the parameter type. If you change the parameter type, the menu returns to [Area copy].

(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 [Area paste]. → The menu will be highlighted and the confirmation message "Paste?(Y/N)" appears.

(3) Press the [Y] or [INPUT] key. → The copied data is written to the parameter with the same number in the area where the cursor is positioned.

When the keys other than [Y] or [INPUT] are pressed, the data will not be written. → The menu highlight returns to normal and the confirmation message "Change the menu back to Area copy?(Y/N)" appears.

<When returning to copying operation>

(4) Press the [Y] or [INPUT] key. → The selected area color of the range to be copied will return to the previous color.
The menu will be changed to [Area copy].

<When continuing to copy>

(4) Press a key other than [Y] or [INPUT]. → The range to be copied remains the selected area color.
The menu remains [Area paste].

(5) Move the cursor to the display area of the axis or the part system to be pasted and then press the menu [Area paste]. → The menu will be highlighted and the confirmation message "Paste?(Y/N)" appears.

(6) Press the [Y] or [INPUT] key. → The copied data is written to the parameter with the same number as the copy source in the area pointed by the cursor.
The operation after this is the same as (3).

Note

(1) If the relevant parameter is changed after specifying the range to be copied, the value after the change is pasted.

7.1.4 Undoing the Parameter Change

This can undo the last rewritten parameter.
The input data can be returned to the previous value. By pressing the menu [Undo] again, the previous value is changed back to the value that you last input.

(1) Input the value in the area pointed by the cursor. → The setting value is displayed and the cursor moves.

(2) Press the menu [Undo]. → The cursor returns to the original position and the value before setting is displayed.

Note

(1) Even when the multiple axes data are set simultaneously using "/", all the data will be returned to the previous values before the setting.

(2) The undo operation remains enabled until the screen or the parameter type is changed.

(3) When the operation parameter "#8939 Undo confirm msg" is set to "1", the menu [Undo] is highlighted and the confirmation message "OK?(Y/N)" appears. The program returns to the previous status by pressing [Y] or [INPUT] key. If any key other than [Y] or [INPUT] is pressed, the operation to undo will be canceled. When the operation is canceled, the menu highlight returns to normal and the message disappears.

7.1.5 Parameter Configuration

The parameter number range and permissible operations for all parameter types are shown below.
○: Can be used Δ: Operable under certain conditions (Refer to the section below for conditions.)

Parameter type	Menu [Next axis]	System change key	Menu [Area copy]	Menu [Area paste]
Fixed cycle parameter	-	Δ (*3)	Δ (*3)	Δ (*3)
Fixed cycle	-	Δ (*3)	Δ (*3)	Δ (*3)
Control parameter 1	-	-	-	-
Control parameter 2	-	Δ (*3)	Δ (*3)	Δ (*3)
I/O parameter	-	-	-	-
Ethernet parameter	-	-	-	-
Computer link parameter	-	-	-	-
Subprogram storage location parameter	-	Δ (*3)	Δ (*3)	Δ (*3)
Axis parameter	Δ (*1)	-	Δ (*2)	Δ (*2)
Barrier data (*4)	-	Δ (*3)	○	○
High-accuracy control parameter	-	Δ (*3)	Δ (*3)	Δ (*3)
High-accuracy control axis parameter	Δ (*1)	-	Δ (*2)	Δ (*2)
Operation parameter	-	-	-	-
Menu selection parameter	-	-	-	-
Tolerance parameter	Δ (*1)	Δ (*3)	Δ (*3)	Δ (*3)
Wireless LAN parameter	-	-	-	-

(*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.

(*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.

(*3) This is valid only when the number of valid part systems is two or more.

(*4) The barrier data appears only for L system.

7.1.6 Echoback

When 13 or more characters are set for Ethernet parameter to be echoed back, the set value is displayed in the input area and displayed in the data column with "...". When the cursor moves to a parameter which is not subject to echoback, the contents in the input area will be deleted.

7.1.7 Ethernet parameter

If the Ethernet parameter "Host 1 password" to "Host 4 password" is set, "********" displays regardless of the password characters set in that data field.

7.1.8 Setting VNC Server Connection Password

If the VNC server connection password is set, "********" is displayed regardless of the number of password setting characters in the data display field.
The details on key operation when inputting the password are saved in the key history as "*".

For details on the VNC server, refer to "17 Appendix 4: VNC Server Function".

7.1.9 Precautions

When the data protection is valid, the value cannot be set (including paste operation) for the high-accuracy control parameters (part system based) and high-accuracy control axis parameters.

7.2 Selecting High-accuracy Control Parameters (Machining Cond I Screen)

Setup – MacCond

You can switch the high-accuracy parameter set to match the machining purpose on object on the "Machining cond" (MacCond) screen of the setup screen. That enables a machining which meets the machining application or machining process. Multiple sets of high-accuracy parameter (parameter related to the high-accuracy control) can be configured in advance for each machining application (such as parts machining or die machining) and machining process (such as rough or finishing).

Procedure for using this function is as follows:

(1) Initializing the machining condition parameter set

- Initializing the machining condition parameter set in the "Machining cond" screen
- Initializing the machining condition parameter set by data input/output

(2) Setting the machining condition parameter set

- Set the total of nine sets of machining condition parameter sets of "three applications x three conditions" in the "Machining cond" screen

(3) Switching the machining condition parameter set

- Switching the machining condition parameter set in the "Machining cond" screen
- Switching the machining condition parameter set by G code command in the machining program
- Switching the machining condition parameter set in the sequence program by a PLC signal

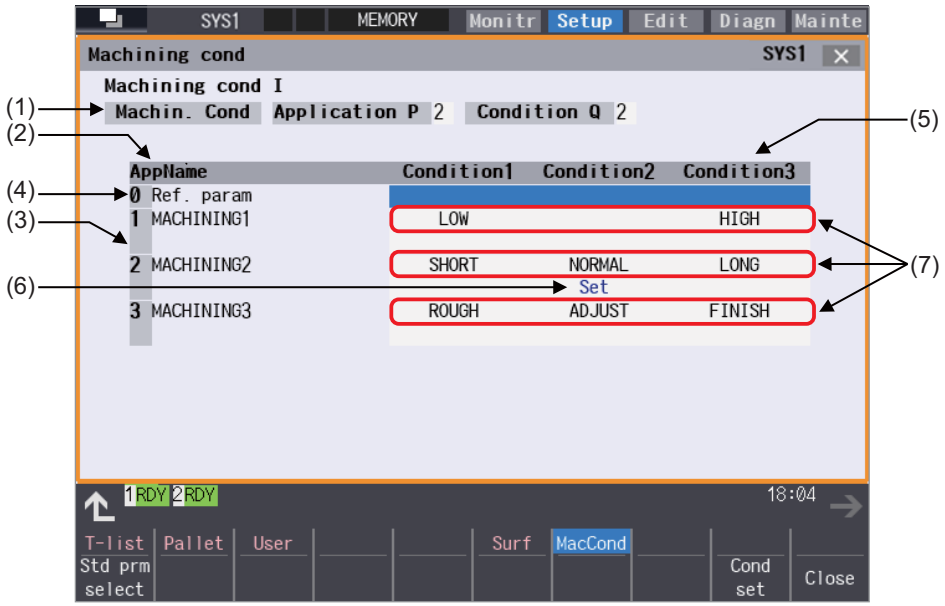
MTB password is required for operating this screen.

Machining condition selection I screen

Display of the machining condition selection screen varies depending on the setting of the parameter "#1760 cfgPR10/bit0" (Machining condition selection screen - Disable setting of condition names).
The cursor position with this function is not retained. When this screen is displayed again after transiting to another screen, the cursor is positioned on the field where "Set" is displayed.

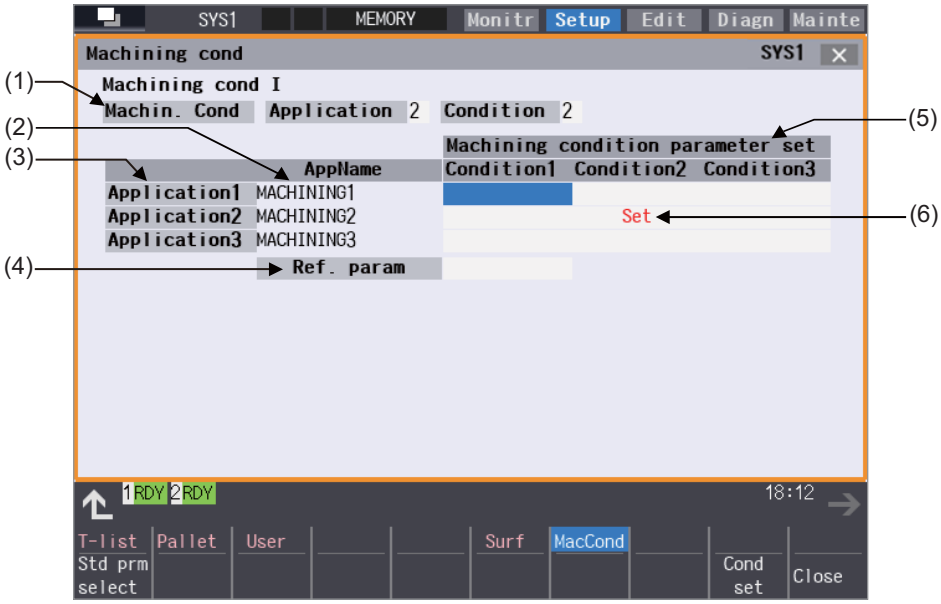
When the parameter "1760 cfgPR10/bit0" is set to "0".

Condition names can be set.



When the parameter "1760 cfgPR10/bit0" is set to "1".

Condition names cannot be set.



Display items

Display items		Details
(1)	During machining condition	Displays the applications and conditions used for the current machining. This shows No. 0 through 3 of the machining applications and conditions, which you have selected on the machining condition selection screen or switched with a G code command or a PLC signal. While standard parameter is being selected, "0" is displayed on the applications and conditions.
(2)	Application name	Displays the application name set in the menu [AppName set] in the "Machining cond screen".
(3)	Application	Displays Application 1 to 3.
(4)	Ref param	"Set" is displayed in this column while the standard parameter is being selected.
(5)	Machining condition parameter set	Displays the machining condition parameter set of application 1 to 3. "Set" is displayed in the column of the selected machining condition parameter set.
(6)	Set	"Set" is displayed in the column of the selected machining condition parameter set on the machining condition selection screen. It will not be updated even if the machining application or condition is switched by a G code command or a PLC signal.
(7)	Condition names	These lines appear when the parameter "1760 cfgPR10/bit0" is set to "0". Condition names set for respective items are displayed. When no condition name is set, the line is blank.

Menus

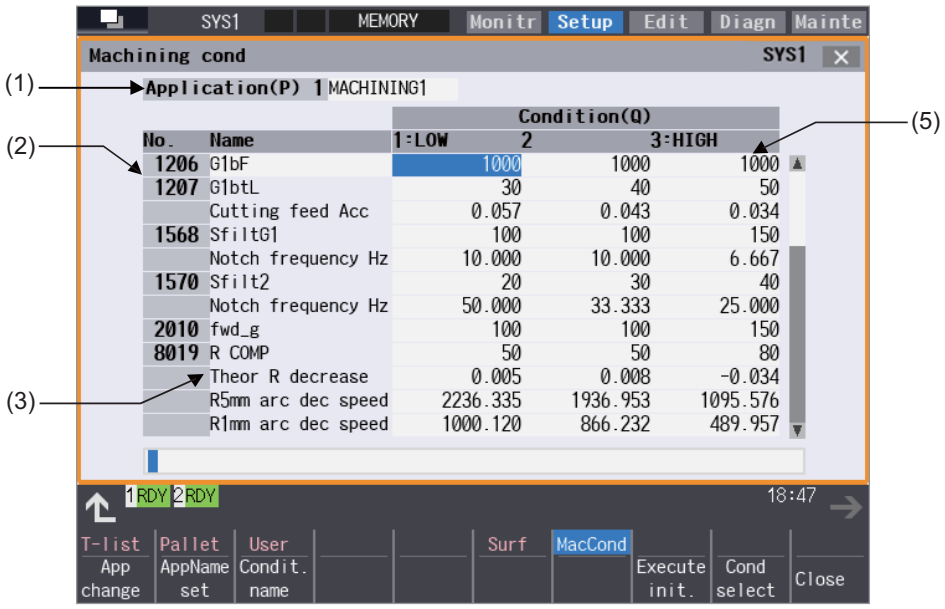
Menus	Details
Std prm select	Switches machining condition to standard parameter. After the menu is highlighted and the cursor moves to standard parameter, the confirmation message "Change to standard parameters?(Y/N)" will appear. If you press [Y] (or [INPUT]), the status of standard parameter changes to "Set". If you press any key other than [Y] (or [INPUT]), it will be canceled. The cursor will remain on standard parameter even when canceled.

Menus	Details
Cond set	Changes to the "Machining cond" screen. <Note> ♦MTB password is required for the maintenance screen.
Close	This closes the pop-up window and quits this function.

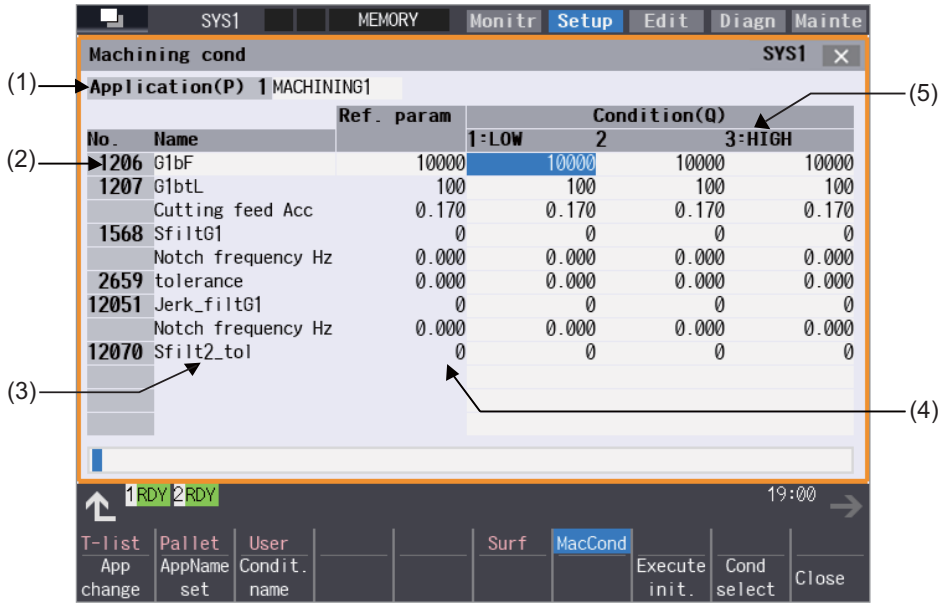
Machining condition setting screen

The machining condition parameter to be displayed is switched by enabling or disabling tolerance control.

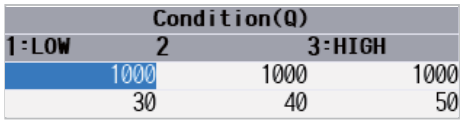
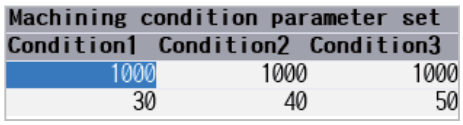
<When tolerance control is disabled>



<When tolerance control is enabled>



Display items

	Display items	Details
(1)	Application	Displays the application being set. To change to the next machining purpose, press the menu [App change]. When machining purpose being selected, the background is emphasized in red. The background is not highlighted while the Standard parameter is selected.
(2)	Parameter No.	Displays the number of a standard parameter.
(3)	Parameter name	Displays the name of a standard parameter.
(4)	Ref param	Displays the value of the standard parameter designated on the parameter screen. This item is not displayed when the machining parameter "#12066 Tolerance ctrl ON" is set to "0". <Note> ♦The parameter "#2659 tolerance" indicates the setting value of the 1st axis in the displayed part system.
(5)	Machining condition parameter set	Displays three sets of machining condition parameter set which belongs to the application purpose being set. The background of the selected machining condition is highlighted in red. The background is not highlighted while the Standard parameter is selected. Display of the title field varies depending on the setting of the parameter "1760 cfg-PR10/bit0". When the parameter is set to "0" (condition names can be set). Display of the title field varies as follows depending on whether condition name is set or not. Condition name is not set: "Number" only Condition name is set: "Number" + ":" + "Condition name"  When the parameter is set to "1" (condition names cannot be set). "Condition1", "Condition2" and "Condition3" are displayed. 

Menus

Menus	Details
App change	Switches the application being set to other application. It switches from Application1 -> Application2 -> Application3 -> Application1 -> … each time this menu key is pressed.
AppName set	Sets the application name for the application being set. The name which consists of up to 12 alphanumeric characters can be set. When "0" is entered, the set condition name is cleared. <Note> ♦"\", "/", ",", "*", "?", "<", ">", "|", and "space" cannot be used.
Condit. name	Sets the condition name for the line where the cursor is located. The name which consists of up to 9 alphanumeric characters can be set. When "0" is entered, the set condition name is cleared. <Note> ♦This menu is not displayed when the parameter "1760 cfgPR10/bit0" is set to "1". ♦"\", "/", ",", "*", "?", "<", ">", "|", and "space" cannot be used.
Execute init.	Copies the setting values corresponding to the standard parameters to the all machining condition parameter sets.
Cond select	Changes to the machining condition selection screen.
Close	This closes the pop-up window and quits this function.

Initializing the machining condition parameter set

(1) Press the menu [MacCond] in the setup screen.

(2) Select the menu [Cond set] in the "Machining cond" screen.

(3) Select the menu [Execute init].

(4) Press the [Y] or [INPUT] key. ➡ The setting value for the standard parameters corresponding to each machining condition parameter will be copied.

Setting the machining condition parameter set

(1) Press the menu [MacCond] in the setup screen.

(2) Select the menu [Cond set] in the "Machining cond" screen.

(3) Move the cursor to the position of the parameter to be set.

(4) Input the value, and then press [INPUT].

Note

(1) It cannot be set without initializing the machining condition parameter set.

Switching the machining condition parameter set

(1) Place the cursor on the position of the machining condition parameter set (application/condition) to be switched in the "Machining cond" screen.

(2) Press the [INPUT] key.

(3) Press the [Y] or [INPUT] key.

Note

(1) The machining condition which is selected on the Machining cond screen applies to all part systems.

(2) When the machining condition parameter set is switched, for the parameter "#2010" (Feed forward gain) and "#2659 Tolerance", the same value is applied to all NC axes in the switched part system.

(3) For the operation search, the machining condition selection parameter is not switched. For the restart search, the machining condition selection parameter is switched.

Canceling the machining condition parameter set and switch to standard parameter (Switch by cursor)

(1) Move the cursor to the standard parameter and press [INPUT] key on the "Machining cond" screen.

(2) Press the [Y] or [INPUT] key.

Canceling the machining condition parameter set and switch to standard parameter (Switch by menu [Std prmselect])

(1) Select the menu [Std prmselect] in the "Machining cond" screen.

(2) Press the [Y] or [INPUT] key.

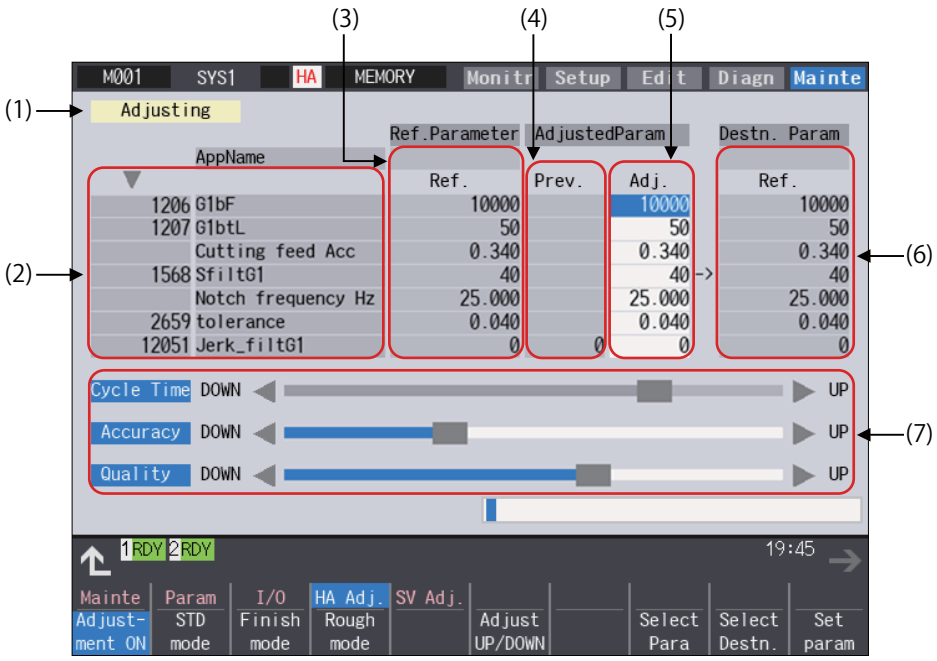
7.3 Adjusting High-Accuracy Control Parameters

Mainte – HA Adj.

High-accuracy control parameters can be adjusted in accordance with the adjustment purpose on the high-accuracy control parameter adjustment screen.
Settings on this screen are only available when the parameter "#12066 Tolerance ctrl ON" is set to "1" (enabled).
Enter the MTB password to select parameter adjustment modes such as rough and finishing.

Adjustment result is set to the reference parameter or machining condition parameter (1-1 to 3-3).

- Use "G120.1 P_ Q_" (machining condition selection I command) to machine with the conditions set in the machining condition parameters.
- "G120.1 P_ Q_" (machining condition selection I command) is not required to machine with the conditions set in the reference parameters.



Display items

	Display items	Details
(1)	Display adjustment mode	This displays the adjustment mode. The message "Adjusting" appears while in adjustment mode.
(2)	High-accuracy control parameter	This is the high-accuracy control parameter set on this screen. It can be scrolled by dragging. <Page status display button> Pages can be changed by pressing the ▲ or ▼ button.
(3)	Reference parameter area	Displays the parameter referenced by the adjustment parameter. Select the parameter to be referenced using [Select Para]. If the standard parameter is selected, [Ref. param] will display. If the machining condition parameter is selected, the following will be displayed: •If no condition name is set: [Cond. 1-1] to [Cond. 3-3] •If a condition name is set: [1-1] to [3-3] + [:] + [condition name] If a machining application name is set, the name will display in the machining application name field. Otherwise, this field will remain blank.

Display items		Details
(4)	Last adjustment parameter area	Displays the previous adjustment parameter area value.
(5)	Adjustment parameter area	Displays the set value for the adjustment parameter. During adjustment, automatic operation is performed with the adjusted parameter area value.
(6)	Configuring parameter area	Displays the destination parameter to be set by the adjustment parameter. Select the parameter to be set using [Select Destn.]. The content displayed here is the same as those given in the "Ref. parameter" section.
(7)	Adjustment bar display	Sets and displays the adjustment conditions (cycle time, accuracy, quality) and their degree. Use the sliders, UP/DOWN buttons, or menu buttons to adjust the degree of adjustment. <Adjustment condition buttons> When an adjustment condition button is selected, that adjustment condition is locked, allowing only the other adjustment conditions to be adjusted. Only one adjustment condition can be locked at a time. <Sliders> The degree of adjustment can be adjusted by sliding the slider button. Unlocked adjustment conditions:  Locked adjustment conditions:  <UP/DOWN buttons> The degree of adjustment can be adjusted by using the  and  buttons.

Menus

Menus	Details
Adjustment ON	Switches the adjustment mode ON and OFF. <Note> •When the adjustment mode is turned ON, the adjustment parameter area is automatically set based on the reference parameter area values, the parameters "#12139 SrvAdjst_Tconst" (Time constant for servo adjustment) and "#12140 MacVib_Tconst" (Machine vibration filter time constant).
STD mode Finish mode Rough mode	Selects a parameter adjustment mode. <Note> •These menu buttons will not display if the MTB password is not input. •In rough mode, only the cycle time can be adjusted. •If either of parameters "#12139 SrvAdjst_TConst" (Time constant for servo adjustment) or "#12140 MacVib_TConst" (Machine vibration filter time constant) is "0", these menu buttons cannot be selected. •When a parameter adjustment mode is selected, the adjustment parameter area is automatically set based on the reference parameter area values, the parameters "#12139 SrvAdjst_Tconst" (Time constant for servo adjustment) and "#12140 MacVib_Tconst" (Machine vibration filter time constant).
Adjust UP/DOWN	Displays the sub-menu for adjusting levels for the adjustment conditions.
Select Para	Displays the sub-menu for reference parameter selection.
Select Destn.	Displays the sub-menu for setting the destination to be set by the adjustment parameter.

Menus	Details
Set param	Sets the adjustment parameter area value to the destination parameter.

Sub-menu buttons under [Adjust UP/ DOWN]

Menus	Details
CY.Time UP	Increments the cycle time for the adjustment parameter up by one step with each press.
CY.Time DOWN	Decrements the cycle time for the adjustment parameter down by one step with each press.
Accur. UP	Increments the accuracy for the adjustment parameter up by one step with each press.
Accur. DOWN	Decrements the accuracy for the adjustment parameter down by one step with each press.
Quality UP	Increments the quality for the adjustment parameter up by one step with each press.
Quality DOWN	Decrements the quality for the adjustment parameter down by one step with each press.
Change Cond.	Switches the locked adjustment condition. Each time the menu button is pressed, this will switch between none locked -> cycle time locked -> accuracy locked -> quality locked -> none locked -> ...

Note

(1) If rough mode is selected for a parameter adjustment mode, the menu buttons for [Accur. UP], [Accur. DOWN], [Quality UP], and [Quality DOWN] will be grayed out.

(2) If an adjustment condition is locked, both the UP and DOWN buttons for that adjustment condition are grayed out.

Sub-menu buttons under [Select Para] and [Select Destn.]

Menus	Details
Ref. param	Selects the parameter set (the standard parameter and machining condition parameters). <Note> ♦The selected parameter set will remain selected even after the power is turned ON again. ♦When a reference parameter is selected with the menu [Select Para], the adjustment parameter area is automatically set based on the reference parameter area values, the parameters "#12139 SrvAdjst_Tconst" (Time constant for servo adjustment) and "#12140 MacVib_Tconst" (Machine vibration filter time constant). ♦If the MTB password is not entered in [Select Destn.], machine conditions with protection enabled cannot be selected.
Cond. 1-1 : Cond. 3-3	

Operation method

(1) Press [HA Adj.] on the maintenance screen.

(2) Press the system change key to switch the current system.

(3) Press [Adjustment ON].
If the MTB password has not been entered, go to step (5).
→ [Adjust-ment ON] will be highlighted, and the message "Adjusting" will appear on the screen.
"HA" will flash on NC status area.

(4) Select a parameter adjustment mode:
STD mode: Adjusts standard parameters, for semi-finishing
Finish mode: For finishing
Rough mode: For rough machining

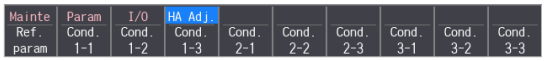
(5) Press [Select Para] and select the criteria for the parameters to adjust.
To adjust based on the reference parameter, select [Ref. param].
→ "Ref." is displayed in the reference parameter area, and the adjustment parameter area is automatically set based on the reference parameter area values.

(6) Use the slider, UP/DOWN buttons, or the [Adjust UP/DOWN] sub-menu buttons to adjust the cycle time, accuracy, and quality in accordance with the adjustment purpose.
Values can also be entered directly by moving the cursor to the parameter to set and adjusting the adjustment values with [↑] and [↓].
→ The adjusted value will be set for "Adj."

(7) Run a machining test.

(8) Repeat steps (6) and (7), adjusting the parameter values until the machining results satisfy the requirements.

(9) Press [Select Destn.] and choose the destination for the adjusted parameter values.
→ The sub-menu buttons for [Select Destn.] is displayed.



To set as the standard parameters, select [Ref. param].

(10) Press [Set param].
→ A message appears confirming whether to set the parameters.

(11) Press the [INPUT] key.
→ The adjusted parameter values are set as the standard parameters.

Precautions

(1) If the standard parameters are selected as the destination parameter, the adjusted parameter values will be propagated to the parameters in the current part system. For the parameter "#2659 tolerance", the same value will be propagated to all NC axes in the current part system.

(2) For the machining test run, do not use a machining program which specifies tolerance amount.

(3) Initialize the machining condition parameters if they are not initialized when the adjustment mode is turned ON by the menu [Adjust-ment ON]. For how to initialize them, refer to "Initializing the Machining Condition Parameter Set" in "7.2 Selecting High-accuracy Control Parameters (Machining Cond I Screen)".

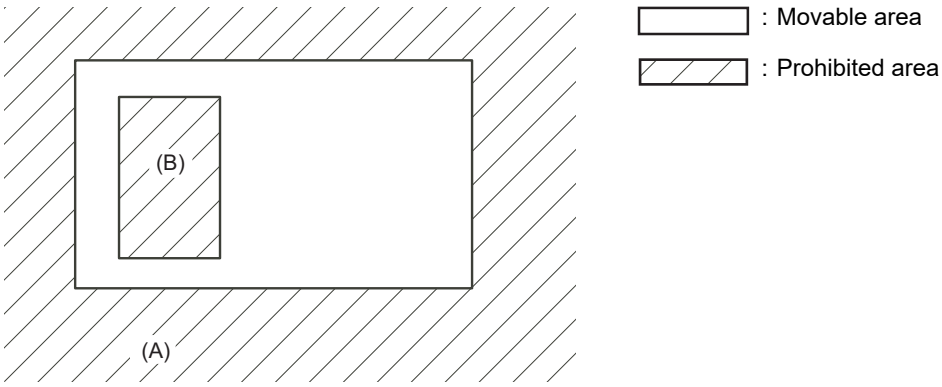
8

Setting Machine Movable Area

This chapter describes the contents which can be set with the user parameters.

8.1 Setting the Stored Stroke Limit

The tool entry prohibited areas can be set with stored stroke limit II or IIB.
Set the parameters to select the entry prohibited area, stored stroke limit II or IIB.



(A): Entry prohibited area by stored stroke limit II (outside)

(B): Entry prohibited area by stored stroke limit IIB (inside)

If the axis is moving over the set area, an alarm will appear and the axis will decelerate to a stop.
If the prohibited area is entered and an alarm occurs, movement will be possible only in the direction opposite the entry direction.

Enabling 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.

Note

(1) 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.
(2) The stored stroke limit is disabled for a sub axis during simple synchronous control.

Stored stroke limit coordinates

The stored stroke limit check is carried out in the basic machine coordinate system established by the reference position return.
When the stored stroke limit has been validated while the reference position return has not been completed, the stored stroke limit check is executed with the basic machine coordinate system at the time of last power-OFF as temporary one.
When the 1st dog-type reference position return is completed after the power is turned ON, the proper coordinate system is established.

⚠ CAUTION

Always set the stored stroke limit. Failure to set this could result in collision with the machine end.

The prohibited areas of the stored stroke limit are set with parameters or program commands.
The upper and lower limits of the area are set with the coordinate values on the basic machine coordinate system for each axis. The stroke check will not be executed when the upper and lower limits are set to the same value.
This function is enabled only for axis whose reference position is established.

Note

(1) Before the axis enters the prohibited area, the operation error (M01 0007) occurs, and the axis will stop moving. The alarm can be reset by moving the erroneous axis in the opposite direction.
(2) During automatic operation, if an alarm occurs with even one axis, all axes will decelerate to a stop.
(3) During manual operation, only the axis that caused the alarm will decelerate to a stop.
(4) The axis will always stop at a position before the prohibited area. The distance between the prohibited area and the stop position depends on the feedrate.

The stored stroke limits II and IIB are handled as follows.

Type	Prohibited area	Area setting parameters	Related parameters
II	Outside	#8204 OT-CHECK-N #8205 OT-CHECK-P	⬧#8202 OT-CHECK OFF: 0 ⬧#8210 OT INSIDE: 0
IIB	Inside	#8204 OT-CHECK-N #8205 OT-CHECK-P	⬧#8202 OT-CHECK OFF: 0 ⬧#8210 OT INSIDE: 1
		#8231 OT-CHECK[2]-N #8232 OT-CHECK[2]-P	⬧#8230 OT-CHECK[2] OFF: 0 ⬧#8210 OT INSIDE: 1

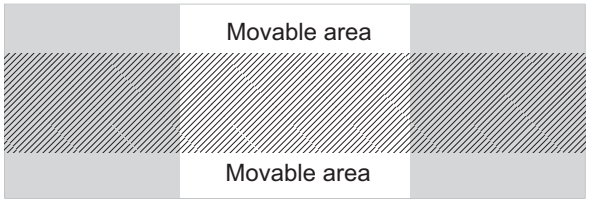
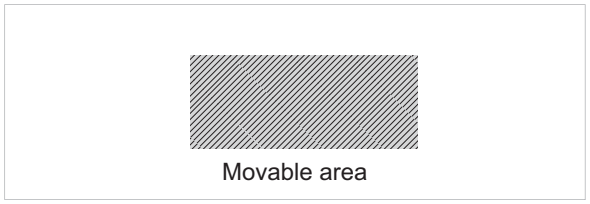
Note

(1) The stored stroke limit is disabled for axis where the reference point is not established.
(2) The stored stroke limit is disabled for axis where the same values are set to the upper and lower limits of the area setting parameters.

8.1.1 Stored Stroke Limit II

The boundary is set with the axis parameters "#8204 OT-CHECK-N" and "#8205 OT-CHECK-P" or with program commands. Either the inside or the outside of the set boundary is the prohibited area. Whether the inside or outside of the area is prohibited is determined by "#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 area is prohibited with G22, and entry into the prohibited area is enabled with G23.
The stored stroke limit II/IIB can be invalidated for each axis with setting "#8202 OT-CHECK OFF" to "1".

When "#8210 OT INSIDE" is set to "1" (IIB enabled), a prohibited area can be set in addition to the area described above. Boundaries of the additional area are set by the axis parameters "#8231 OT-CHECK[2]-N" and "#8232 OT-CHECK[2]-P". The inside of the set boundary is the prohibited area. The boundaries of the additional area cannot be set with the program commands (G22/G23).
The stored stroke limit IIB function of the additional area can be invalidated for each axis by setting "#8230 OT-CHECK[2] OFF" to "1".
When "#8210 OT INSIDE" is set to "0" (II enabled), the stored stroke limit IIB function of the additional area is disabled. The setting values of "#8230 OT-CHECK[2] OFF", "#8231 OT-CHECK[2]-N" and "#8232 OT-CHECK[2]-P" are ignored.

Horizontal axis: X Vertical axis: Z		Movement prohibited area
X	Outside prohibited	 (*1)
Z	Outside prohibited	
X	Outside prohibited	 (*1)
Z	Inside prohibited	
X	Inside prohibited	 (*2)
Z	Inside prohibited	

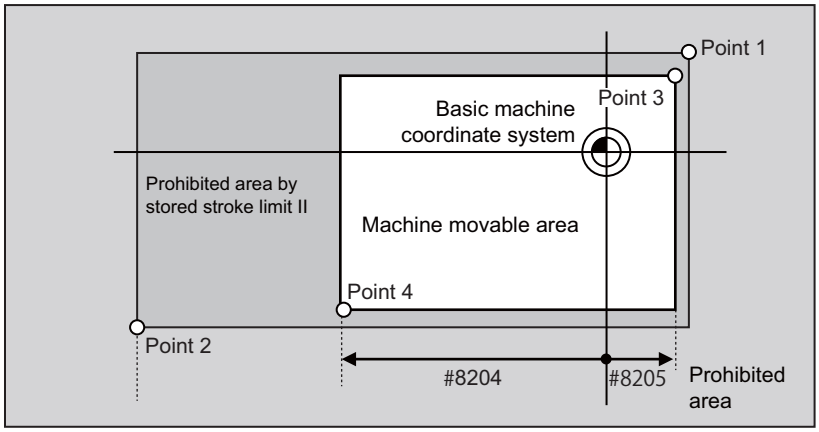
(*1) When the outside area is prohibited for even one of X and Z axes, the prohibited area is the entire area specified for the X and Z axes.

(*2) When the inside area is prohibited for both X and Z axes, the prohibited area is the overlap of the areas specified for the X and Z axes.

(1) Stored stroke limit II (When prohibited area is on outside)

When this function is used with the stored stroke limit I, the tolerable area by both functions is the movable area.

The stored stroke limit I is set by MTB.



The following areas prohibited by stored stroke limit II are set with the coordinate values in the basic machine coordinate system.

Point 3: #8205 OT-CHECK-P

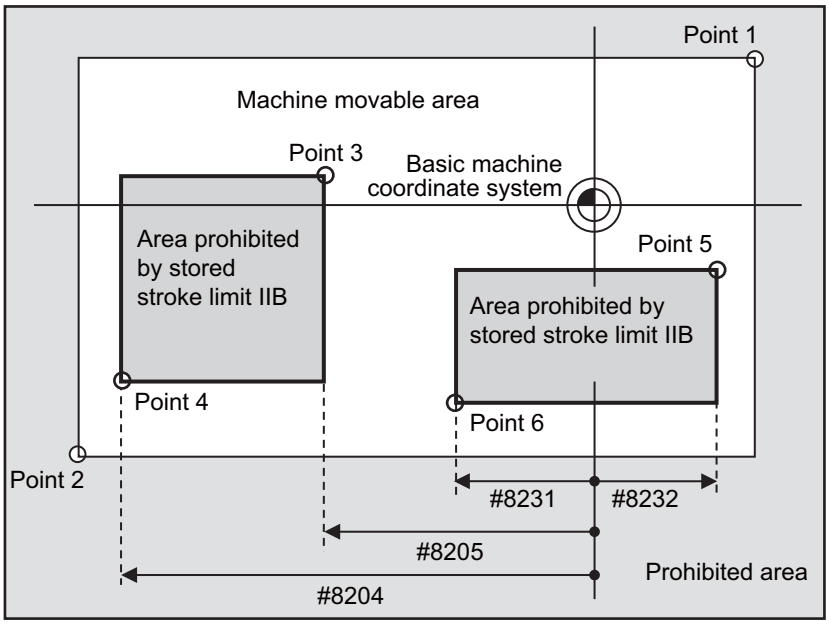
Point 4: #8204 OT-CHECK-N

The area determined by points 1 and 2 is the prohibited area set by MTB using the stored stroke limit I.

(2) Stored stroke limit IIB (When prohibited area is on inside)

An area except for that of the stored stroke limit I becomes the movement prohibited area.

The stored stroke limit I is set by MTB.



The following areas prohibited by stored stroke limit IIB are set with the coordinate values in the basic machine coordinate system.

Point 3: #8205 OT-CHECK-P

Point 4: #8204 OT-CHECK-N

Point 5: #8232 OT-CHECK[2]-P

Point 6: #8231 OT-CHECK[2]-N

The area determined by points 1 and 2 is the prohibited area set by MTB using the stored stroke limit I.

8.2 Chuck Barrier/Tailstock Barrier (L System)

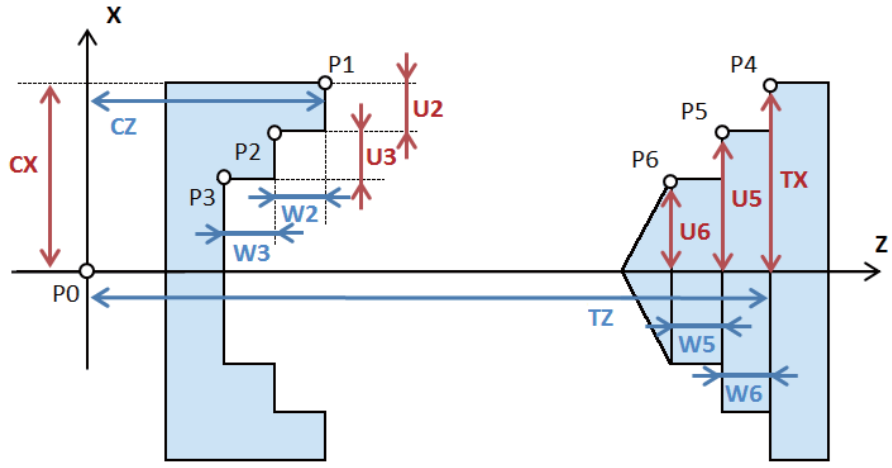
Setup – Barrier

Use this function to set and display the chuck barrier and tailstock barrier.
By limiting the tool nose movement range, the chuck barrier and tailstock barrier prevent the tool from colliding with the chuck and tailstock due to programming errors.
Three points can be input for both the chuck barrier and tailstock barrier. Select either [Enter length] or [Enter coordin] to input (set) the barrier data.

(1) Enter length: To set the barrier position of the chuck or tailstock with the length (length between P points).

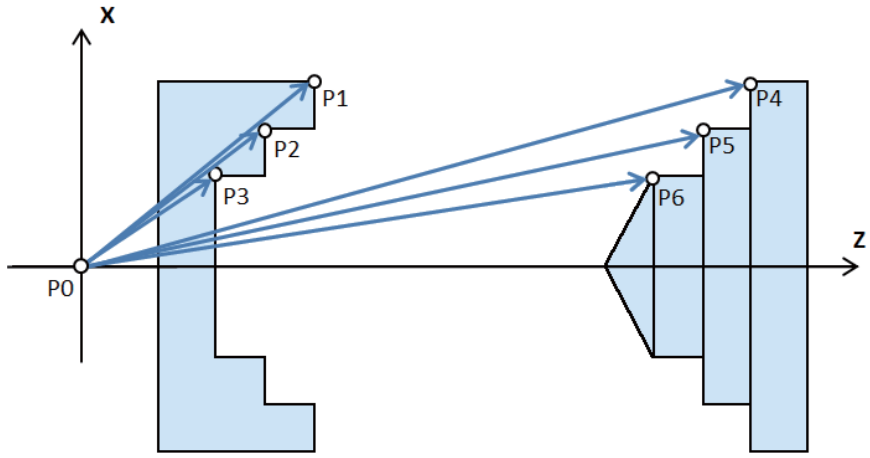
(2) Enter coordin: To set the barrier positions of the chuck and tailstock with the distance from the workpiece center coordinates (P0). Input the radius value to set the X axis.

(1) Enter length



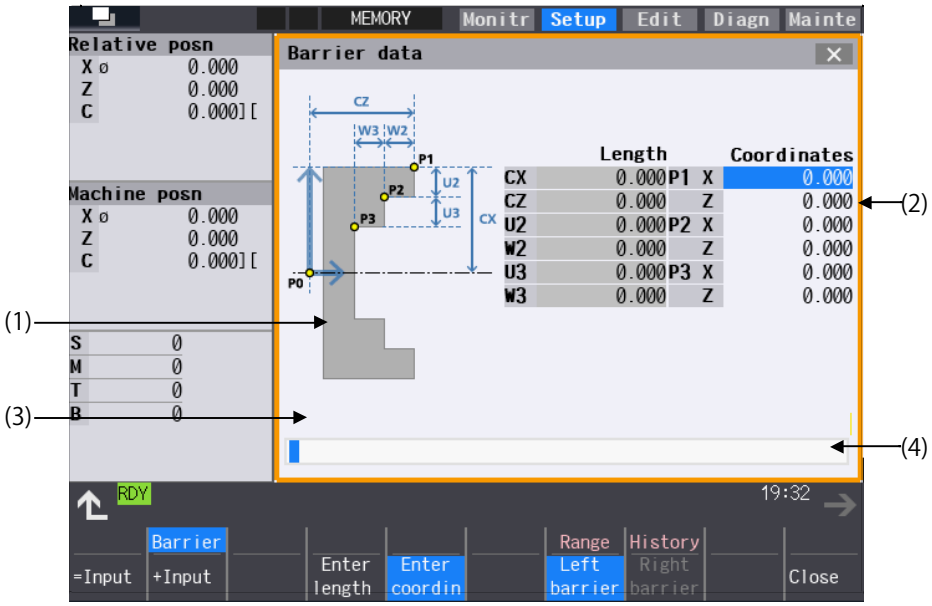
<Chuck barrier> <Tailstock Barrier>

(2) Enter coordin

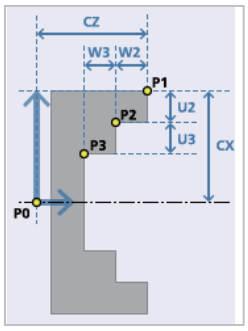
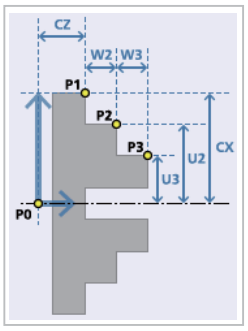
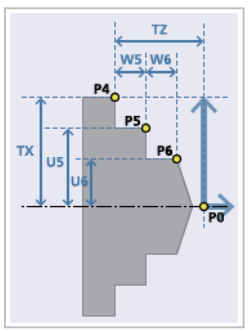
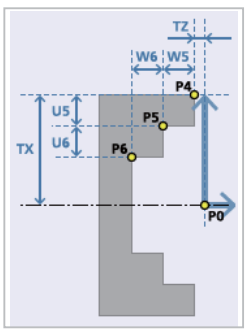


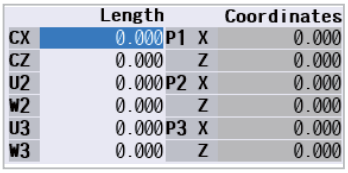
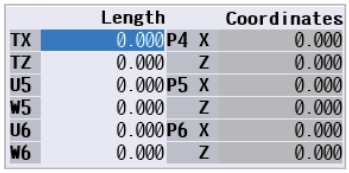
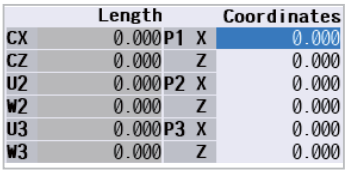
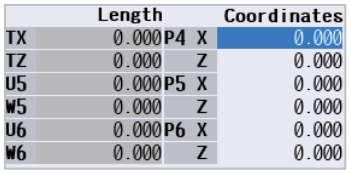
<Chuck barrier> <Tailstock Barrier>

The barrier data which is set on this screen will be set to NC by converting to the parameter "#8301 P1" to "#8306 P6". Specify the workpiece center coordinates (P0) of X axis for the chuck and tailstock in advance with the parameter "#8300 P0" on the menu [Barrier data] of parameter screen. The workpiece center coordinates (P0) of Z axis are the machine co-ordinates zero point.



Display items

	Display items	Details
(1)	Guide graphic	This displays the guide drawing of barrier type. The guide drawing is switched according to the selected barrier form. The drawing is the left-side barrier guide drawing. Chuck type 1 guide drawing  Chuck type 2 guide drawing  Tailstock type 1 guide drawing  Tailstock type 2 guide drawing 

Display items	Details
(2) Barrier data	This displays the barrier data. The display items differ depending on the length/coordinates value input and left/right-side barrier. The setting range is from -99999.999 to 99999.999 (mm). However, minus data cannot be input for length data. <Length input mode> Set the chuck or tailstock position with the length (length between P points). When the value is set, the coordinates data is also updated. Chuck barrier  Tailstock barrier  <Coordinates input mode> Set the chuck or tailstock position with the distance from the workpiece center coordinates (P0). When the value is set, the length data is also updated. Chuck barrier  Tailstock barrier 
(3) Error message	This is displayed when barrier data setting is illegal.
(4) Input section	This displays the input data.

Menus

Menus	Details
=Input	This executes an absolute input.
+Input	This executes an incremental input.
Enter length	This changes to the length input.
Enter coordin	This changes to the coordinates value input.
Left barrier	This changes to the left-side barrier data setting. When the parameter "#8315 BARRIER TYPE (L)" is "0", this menu cannot be selected.
Right barrier	This changes to the right-side barrier data setting. When the parameter "#8316 BARRIER TYPE (R)" is "0", this menu cannot be selected.
Close	This closes the pop-up window and quits this function.

Menus when the menu [Left barrier] or [Right barrier] is pressed

Menus	Details
Chuck type 1	This changes the barrier form to the chuck barrier type 1. (*1)
Chuck type 2	This changes the barrier form to the chuck barrier type 2. (*1)
T-stock type 1	This changes the barrier form to the tailstock barrier type 1. (*2)
T-stock type 2	This changes the barrier form to the tailstock barrier type 2. (*2)

(*1) This menu can be selected when the parameter "#8315 BARRIER TYPE (L)" or "#8316 BARRIER TYPE (R)" is "1".

(*2) This menu can be selected when the parameter "#8315 BARRIER TYPE (L)" or "#8316 BARRIER TYPE (R)" is "2".

Setting the barrier position with length (length between P points)

(1) Press the menu [Enter length]. → The mode is switched to the length input mode. The highlight of the menu [Enter coordin] is canceled and the [Enter length] is highlighted.
The cursor is displayed at CX, and the length data can be set.

(2) Select whether the barrier to be set is right side or left side with the menu.
(Example) Press the menu [Left barrier]. → The display is switched to the menu of the barrier form.

(3) Select the chuck type with the menu.
(Example) Press the menu [Chuck type 2]. → The drawing is switched to the chuck barrier type 2 guide drawing.

(4) Input the value to CX.
20.000 [INPUT]
The setting can be performed also by pressing the menu [=Input] or [+Input] instead of the [INPUT] key. When the menu [+Input] is pressed, the value obtained by adding the input value to the original value is set. → The value "20.000" is set to the cursor position (CX), and the coordinates value is also updated.
The cursor moves to CZ.
Consistency check is performed for respective setting data, and an error message is displayed when an illegality is detected.

(5) Input the values to CZ to W3 in the same way.

Setting the barrier position with coordinates value (distance from the work center coordinates (barrier P0))

(1) Press the menu [Enter coordin]. → The mode is switched to the coordinates input mode. The highlight of the menu [Enter length] is cancelled and the menu [Enter coordin] is highlighted. The cursor is displayed at P1 X, and the coordinates data can be set.

(2) Select whether the barrier to be set is right side or left side with the menu.
(Example) Press the menu [Right barrier]. → The display is switched to the menu of the barrier form.

(3) Select the tailstock type with the menu.
(Example) Press the menu [T-stock type 1]. → The drawing is switched to the tailstock barrier type 1 guide drawing.

(4) Input the value to P1 X.
30.000 [INPUT]
The setting can be performed also by pressing the menu [=Input] or [+Input] instead of the [INPUT] key. When the menu [+Input] is pressed, the value obtained by adding the input value to the original value is set. → The value "30.000" is set to the cursor position (P1 X), and the length is also updated. The cursor moves to P1 Z. Consistency check is performed for respective setting data, and an error message is displayed when an illegality is detected.

(5) Input the values to P1 Z to P3 Z in the same way.

Precautions

(1) When the parameter "#8310 Barrier ON" is set to "0", the chuck and tailstock barriers settings are disabled.

(2) The parameter "#8300 P0" and "#8311 P7" to "#8314 P10" cannot be set on this screen. Set the parameters with the menu [Barrier data] on the parameter screen.

(3) When the parameters "#8315" and "#8316" are both "0", the barrier data is handled as left barrier and chuck type 1. The chuck type cannot be changed.

(4) When the parameter for chuck or tailstock is set to both right and left barriers, an error message appears.

(5) When multiple errors are occurred, an error concerning the X axis is preferentially displayed.

9

Other Setups

9.1 Pallet Program Registration

Setup – 

Machining programs can be registered per each pallet of the automatic pallet changer (hereinafter APC) and the indexing plane from the menu [Pallet] on the setup (Setup) screen.

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.

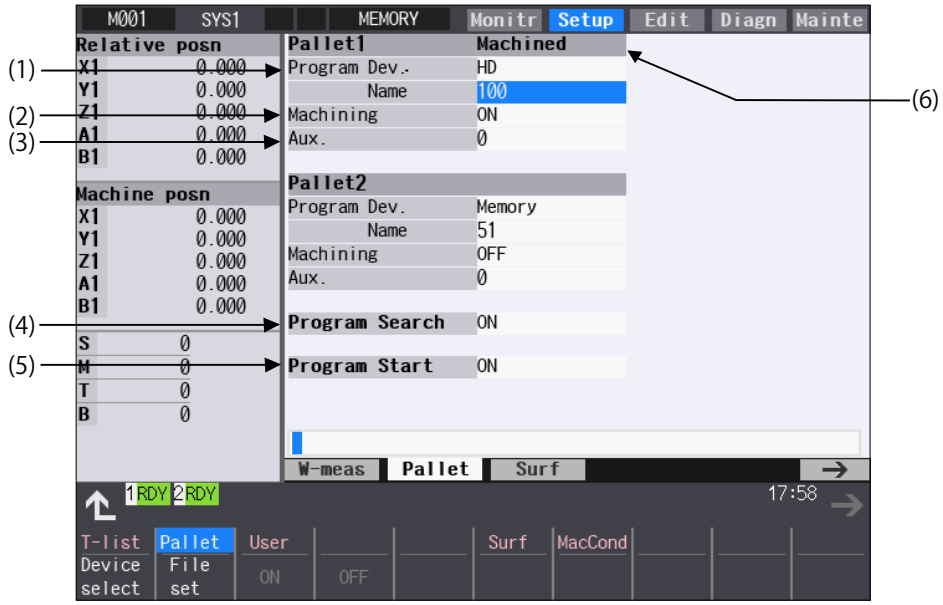
#11001APC type	Screen		Function
0	Standard Pallet Registration		Registration of machining programs for pallet 1 & 2 is carried out. Pallet 1 & 2 can be registered at once.
1	Pallet 4 Page Registration screen	Pallet List Screen	Machining program device, file name, machining status, and index planes under 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

(1) The number of pallets is determined by the parameter "#11002 Valid pallet num".

Manual numerical value command can be executed when manual numerical value command window is displayed by inputting an address key such as MST, etc.
For the pallet 4 page registration screen, the screen type (pallet list and pallet details) is memorized. When the screen is transferred to other screen and displayed again, the type which was displayed a moment ago is displayed. For the file selection, the pallet details is displayed.

9.1.1 Standard Pallet Registration



Display items

	Display items	Details
(1)	Pallet, Device, File 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)	Auxil.	If auxiliary function is available, the auxiliary function data for each pallet is displayed. Refer to the manual issued by the MTB for details on the corresponding functions.
(4)	Program Search ON/OFF	Whether search for the registered machining program is valid or invalid is displayed.
(5)	Program Start ON/OFF	The ON/OFF state of continuous start for the registered machining program 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)	Machined	"Machined" is displayed for the pallet in the machine.

Menus

Menus	Details
Device select	This displays the device menu. When this menu is selected, the device name of the pallet where the cursor is currently located is changed. When a device other than NC memory or NC memory 2 is selected, the root is displayed first.
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.
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".
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".

Sub-menus of [Device select]

Menus	Details
Memory	When the device menu is selected, the device name of the pallet where the cursor is currently located is changed. The menu [HD] is not displayed on M800VS/M80V Series. (*1)
Memory2	
HD	
Memory card	
DS	
USB Memory	

(*1) When the HD mode operation on an industrial computer (IPC) is enabled with M80V Series, the menu [HD] becomes selectable and you can select files in the hard disk on the industrial computer. The availability of this function depends on the MTB specifications (parameter "#1760 cfgPR10/bit2").
Set the IP address of the industrial computer to the parameter #11005 to use this function.

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 [Device select] and select a device. → The cursor moves to "Program Dev." column. The device menus are displayed. After selecting any of them, the selected device is displayed in the "Program Dev." column.

(3) Move the cursor to "Program Name" and input the program No.

(4) Move the cursor to "Machining" and press the menu [ON] or [OFF].

(5) Move the cursor to "Program Search" and press the menu [ON] or [OFF].

(6) Move the cursor to "Program Start" and press the menu [ON] or [OFF].

Operation method (Selecting a file from the list)

(1) Move the cursor to "File name" and press the menu [File set].

(2) Move the cursor to an arbitrary file and press the [INPUT] key.
(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.

(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.

(3) If a directory is specified, a setting error occurs.

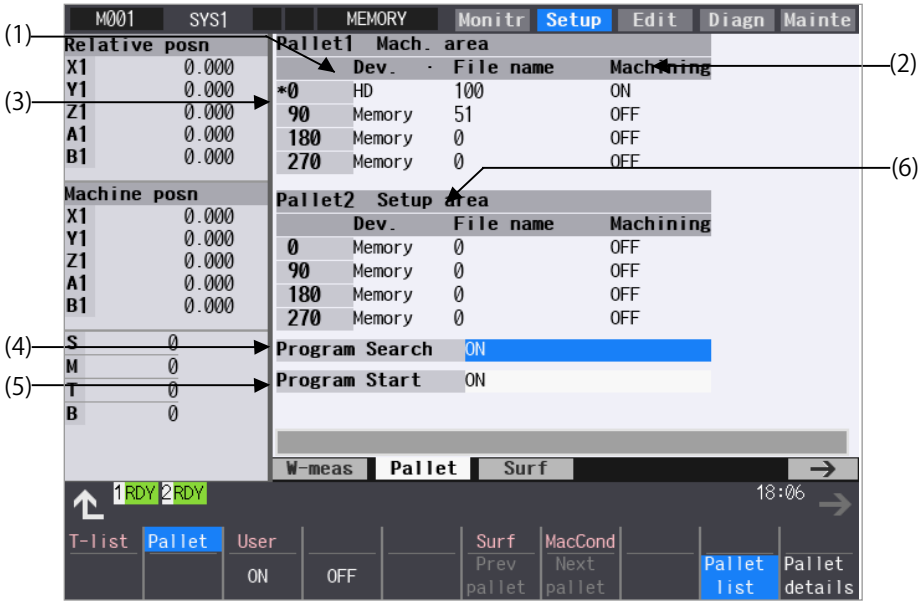
(4) When the menu [Sort change] is pressed, the sorting method is changed. For details, refer to "10.1.3 Changing the Sorting Method".

9.1.2 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

(1) The number of pallets is determined by the parameter "#11002 Valid pallet num".



Display items

Display items		Description
(1)	Pallet, Device, 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	The ON/OFF state of search for the registered machining program 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 system 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	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 valid.
(6)	Pallet information in the machine	Whether pallet is in the machine is displayed. •Mach. area: When pallet is in the machine •Setup area: When pallet is not in the machine

Menus

Menus	Description
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.
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.
Prev pallet	Information on the previous two pallets is displayed.
Next pallet	Information on the next two pallets is displayed.
Pallet list	A list of pallets is displayed.
Pallet details	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.

Operation method (Changing display pallet)

(1) When the pallet 1 and 2 are displayed, press the menu [Next pallet] or next page key. → Pallet 3 and 4 are displayed.

(2) Press the menu [Prev pallet] 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 menu [ON] or [OFF]. → 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 menu [ON] or [OFF]. → The contents set in "Program Start" is displayed. The cursor moves to "Program Search".

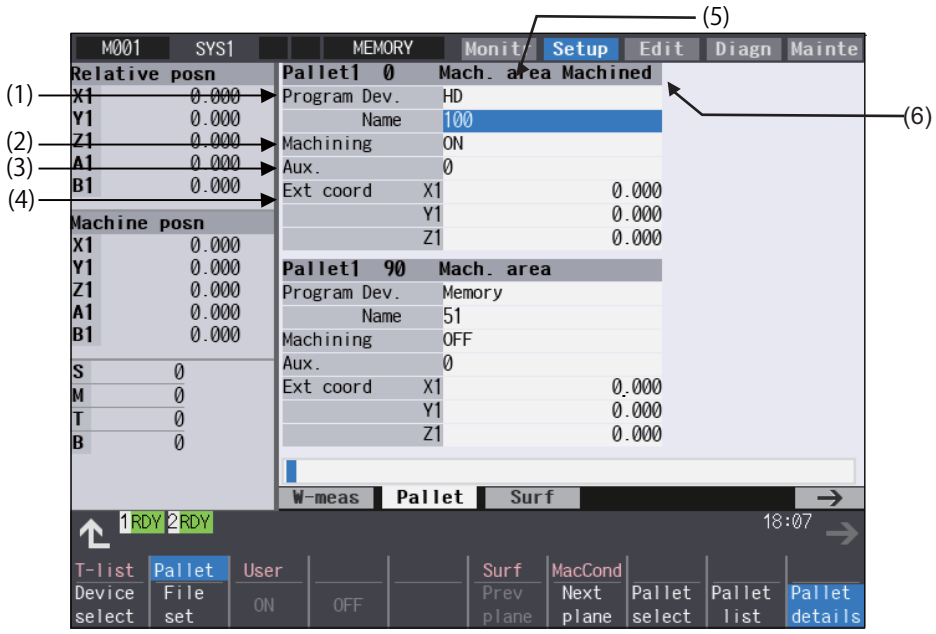
Every time the [↑], [↓], [|←] or [→|] key is pressed, the cursor switches between "Program Search" and "Program Start".

9.1.3 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

(1) The number of pallets is determined by the parameter "#11002 Valid pallet num".



Display items

	Display items	Description
(1)	Pallet, Device, 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)	Aux.	If auxiliary function is available, the auxiliary function data for each pallet is displayed. Refer to the manual issued by the MTB for details on the corresponding functions.
(4)	Ext coord	The set data (X to Z axis) is set in the pallet No. and extended workpiece coordinate offset corresponding to the index plane. The value set in the extended workpiece coordinate offset is written into the external workpiece coordinate system shift (EXT) according to the command from PLC. When the extended workpiece coordinate offset for the machining surface of a pallet in machine is changed, only the changed axis data is written into the external workpiece coordinate system shift (EXT). X to Z axis: External workpiece coordinate data (Example) 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
(5)	Pallet information in the machine	Whether pallet is in the machine is displayed. •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) "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.

(2) For the pallet 4 page registration screen, do not use the extended workpiece coordinate offset because the extended workpiece coordinate offset value is reflected on the external workpiece coordinate system shift.

Menus

Menus	Description
Device select	This displays the device menu. When selected, the device name of the pallet currently indicated by the cursor is changed. When a device other than NC memory or NC memory 2 is selected, the root is displayed first.
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.
ON	The state (Machining) indicated by the cursor is validated. This menu can be used only when the cursor is at Machining.
OFF	The state (Machining) indicated by the cursor is invalidated. This menu can be used only when the cursor is at Machining.
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.
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.
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 is displayed. The cursor will move to the 0° machining program file name.
Pallet list	The Pallet list screen will open.
Pallet details	This displays the pallet details.

Refer to "9.1.1 Standard Pallet Registration" for details of the sub-menus displayed by pressing the menu [Device].

Operation method (Changing display pallet)

(1) Press the menu [Pallet select]. Input pallet No. and press the [INPUT] key.
(Example) 5 [INPUT]
→ The pallet 5's 0° and 90° index plane will appear.

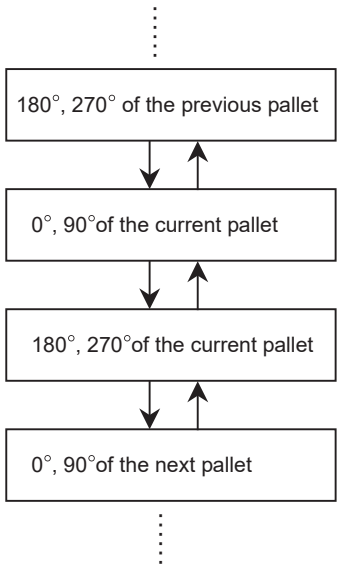
Operation method (Changing index planes)

(Example) When the screen display status acquired after the above (Changing display pallets) operation

(1) Press the menu [Next plane] or next page key. → The pallet 5's 0° and 90° 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 [Device select] and select a device. → The cursor moves to "Program Dev." column. The device menus are displayed. After selecting any of them, the selected device is displayed in the "Program Dev." column. Then the cursor moves to "Name" column.

(3) Move the cursor to "Program Name" and enter the program No.

(4) Move the cursor to "Machining" and press the menu [ON] or [OFF].

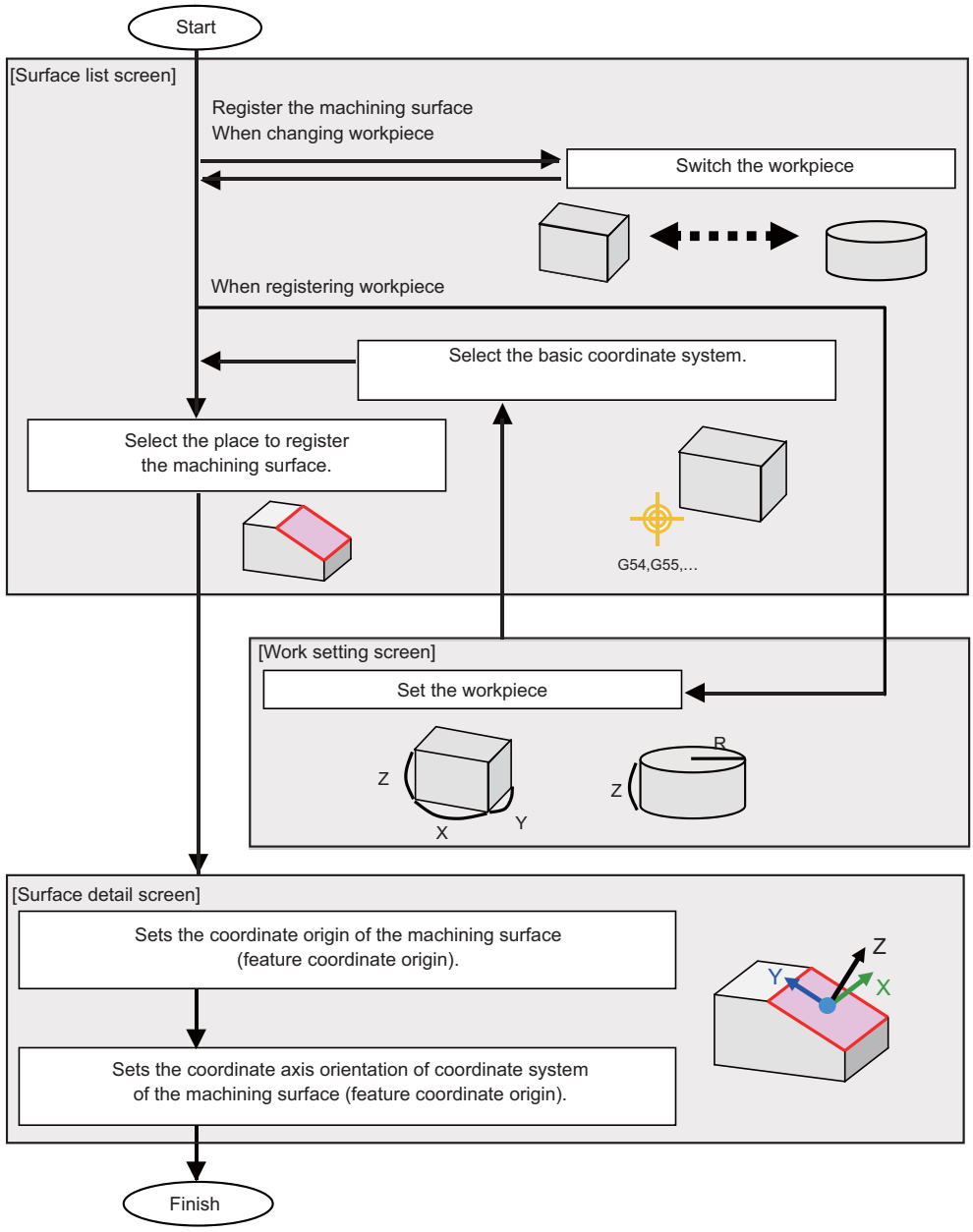
(5) Move the cursor to "Ext coord" and input values for 3 axes.

9.2 R-Navi

Setup – Surf

By setting machining surface coordinate system (feature coordinate system), an optional machining surface can be registered. By selecting registered machining surface on surface selection of monitor screen, commands programmed along three orthogonal axes can move the tool along the coordinate system of this machining surface (feature coordinate system). This selected surface is called a machining target surface. This enables a program with three orthogonal axes to carry out multi-surface machining including inclined surface machining.
This function is an optional specification, and the setting is common to all the part systems.

The following flow chart shows the sequence of registering a machining surface.



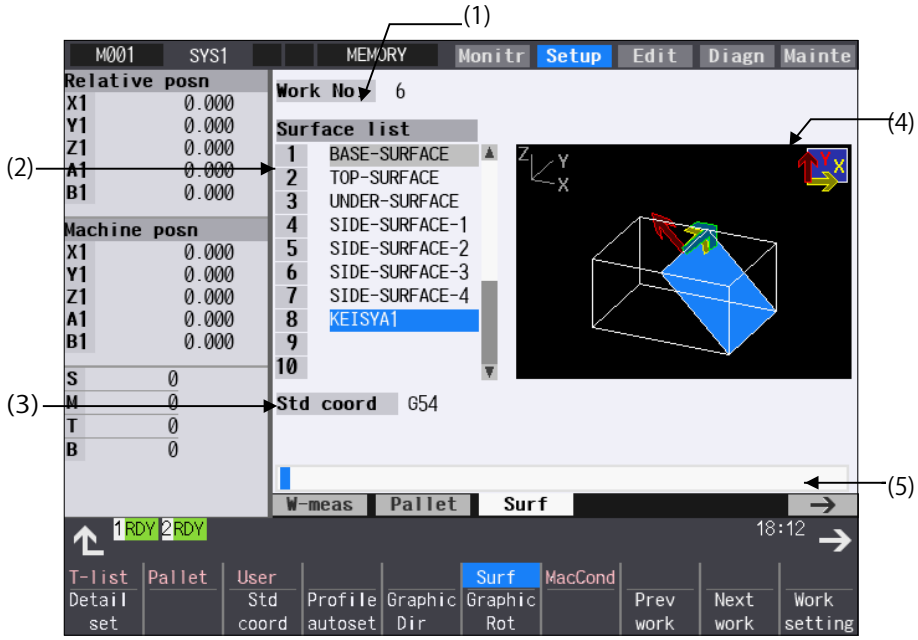
9.2.1 Selecting where to Register a Machining Surface (Machining Surface List Screen)

On the surface list screen the place to register a machining surface can be selected.

Note

(1) When workpiece is initially registered, the surface "BASE-SURFACE" registers. The coordinate zero and axial directions of this machining surface is the same as of the basic coordinate system.
The following operations are disabled for this machining surface. The message "Setting error", "Clear not possible", or "Paste error" is displayed.

- Change the surface name
- Clear the surface
- Paste the surface
- Display the data (coordinate zero/axial direction) of the surface using the menu [Detail set]



Display items

	Display items	Description
(1)	Work No.	Displays the workpiece to which a machining surface is registered. The workpiece size, etc. are set through the work setting screen which is opened by the menu [Work setting].
(2)	Surface list	Displays a list of machining surfaces registered to the workpiece. Besides BASE-SURFACE up to 16 surfaces can be registered in this list.
(3)	Basic coordinate system	Displays the basic coordinate system of the workpiece (workpiece coordinate system G54, G55...).
(4)	Workpiece graphic	Displays the workpiece and the registered surfaces in a graphic form. Displays the workpiece and the registered surfaces in a graphic form. The surface pointed by the cursor in the list is highlighted, and the X, Y and Z coordinate axes of the surface (feature coordinate system) are indicated with arrow marks. (X axis is in yellow, Y in red and Z in green)
(5)	Input section	Displays the key entered. The data entered here will be reflected in the current cursor position when the [INPUT] key is pressed.

Menus (T code list search window)

Menus	Description
Detail set	This opens the surface detail screen as a pop-up window. The origin and axial directions of the feature coordinate system defined on the selected surface can be set on the surface detail screen.
Std coord	This selects the basic coordinate system of the workpiece that is subject to surface registration (G54 to G59 and extended workpiece coordinate G54.1Pn). After selecting, the workpiece coordinate system is displayed as the basic coordinate system. Pressing this switches to the menu [Std coord].
Profile autoset	If this menu is pressed while the workpiece subject to surface registration is a block, the top, bottom and four side faces of the workpiece are automatically registered as machining surfaces. The coordinate systems (feature coordinate systems) of each surface are also registered automatically.
Graphic Dir	This changes the coordinate system direction of the workpiece graphic (graphic display of the surface). If the coordinate system direction is switched using this menu, it is reflected on the workpiece graphic on the details/work setting screen, and held even after the power is turned OFF and ON.
Graphic Rot	This switches the workpiece graphic (graphic display of the surface) by rotating it 90 degrees at a time around the vertical direction. The rotation by this menu is reflected on the workpiece graphic on the details/work setting screen, and held even after the power is turned ON again.
Prev work	This changes the workpiece subject to surface registration to the previous workpiece. If there is no previous workpiece, the last workpiece is displayed.
Next work	This changes the workpiece subject to surface registration to the next workpiece. If there is no next work, the first work is displayed.
Work setting	This opens the work setting screen as a pop-up window. On this screen, multiple workpieces can be registered (name registration), and the workpiece form (type and size) and position can be set to each of the registered workpieces.
Surface copy	This copies the surface data of the specified line (one line). (*1)(*3)(*4)(*5) If the [INPUT] key is pressed without specifying the line, the line where cursor is displaying will be copied.
Surface paste	This writes the copied surface data (one line) to the line where the cursor is displaying. (*1)(*2)(*3)(*5)
Surface clear	This clears the surface data of the specified line (multiple lines possible). (*5)(*6) Methods: Specify the No. of surface to start and the No. to finish deleting. (Example) 1/E: Delete all surface data. The line where the cursor is displaying will be deleted if the [INPUT] key is pressed without specifying the line.

If the size has not been set for the workpiece, the menus except for [Prev work], [Next work], and [Work setting] will be grayed out and non-selectable.

(*1) If the surface data of the copied line is changed, the data will be revised when it is pasted.

(*2) This operation is disabled for the machining target surface (a surface along which three orthogonal coordinate axes are controlled at present) or the surface "BASE-SURFACE". The following message appears.
Machining target surface: "Can't paste machining surface"
Surface "BASE-SURFACE": "Paste error"

(*3) If the surface data of the copied line is deleted, pasting the data will be disabled. The message "Paste error" is displayed.

(*4) The copied line will be maintained even if the screen changes.

(*5) When the data protection is valid, this operation will be disabled. The message "Data protect" is displayed.

(*6) When a line is deleted, this operation is disabled for the machining target surface (a surface along which three orthogonal coordinate axes are controlled at present) the surface "BASE-SURFACE". The following message appears.
Machining target surface: "Can't clear machining surface"
Surface "BASE-SURFACE": "Clear not possible"

If the range specified for this operation includes BASE-SURFACE or the machining target surface (a surface along which three orthogonal coordinate axes are controlled at present) (1/E, for example), the surface data except for the surface "BASE-SURFACE" and the machining target surface will be deleted.

9.2.1.1 Selecting Where to Register a Machining Surface

Operation method

(1) Move the cursor to the line where a machine surface is registered by using arrow keys ([↑], [↓]).

(2) Input the surface name. → The surface name is registered.

When not registering a surface name, skip this and go to operation (3).

(3) Select the menu [Detail set] or press the [INPUT] key. → This opens the surface detail screen as a pop-up window.

Note

(1) Set the surface name with up to 15 one-byte characters (numbers, upper-case alphabets and the symbols that the system recognizes). If a one-byte lower-case alphabet is used for naming, it will be changed to the upper-case letter. Note that the following characters cannot be used.
\ / : , * ? " < > | space
A character string made of a single-byte zero only cannot be registered as a work name.

(2) The operation cannot be operated if the size is not set to workpiece. The message "Setting error" is displayed.

(3) When the data protection is valid, the surface name cannot be changed. The message "Data protect" is displayed.

9.2.1.2 Changing the Surface Name

Operation method

(1) Move the cursor to the row where the name of machine surface needs to be changed using arrow keys ([↑], [↓]).

(2) Input the surface name.
To clear the surface name, input "0" (one-byte zero). → The surface name is registered.

Note

(1) When this operation is operated to the surface "BASE-SURFACE", the message "Setting error" is displayed.

(2) This operation cannot be operated to the machining target surface (a surface along which three orthogonal coordinate axes are controlled at present). The message "Can't set machining surface" is displayed.

(3) When the data protection is valid, the surface name cannot be changed. The message "Data protect" is displayed.

9.2.1.3 Switching the Workpiece

Operation method

(1) Press the menus [Prev work] and [Next work].

<When the menu [Prev work] is pressed>
This will change the workpiece subject to surface registration to the previous workpiece. If there is no previous workpiece, the last workpiece is displayed. (The workpiece that has not set the size is displayed, too.) The cursor moves to the top line.

<When the menu [Next work] is pressed>
This will change the workpiece subject to surface registration to the next workpiece. If there is no next work, the first work is displayed. (The workpiece that has not set the size is displayed, too.) The cursor moves to the top line.

9.2.1.4 Selecting the Basic Coordinate System

The basic coordinate system of the workpiece subject to surface registration can be changed by selecting the menu [Std coord] and then selecting the sub-menu to be displayed.

Operation method

<Selecting the coordinate system G54-G59>

(1) Press the menu [Std coord].
The screen returns to the main menu of the surface list screen when [Return] is selected.
→ The sub-menu is displayed.

(2) Press the menu [G56].
→ G56 is set to the basic coordinate system.
The screen returns to the main menu of the surface list screen.

<Selecting the coordinate system G54.1P>

(1) Press the menu [Std coord].
→ The sub-menu is displayed.

(2) Select the menu [G54.1P].
Highlight of the menu will be canceled by selecting the menu [Return].
The screen returns to the main menu of the surface list screen by selecting [Return] again.
→ The message "Input P number" is displayed.
The menu is highlighted.

(3) Input "20" and press the [INPUT] key.
→ G54.1P20 is set to the basic coordinate system.
The screen returns to the main menu of the surface list screen.

Note

(1) The basic coordinate system of the workpiece to which the machining target surface (a surface along which three orthogonal coordinate axes are controlled at present) is registered cannot be changed. The message "Can't set work including mach surface" is displayed.

(2) When the data protection is valid, the basic coordinate system cannot be changed. The message "Data protect" is displayed.

9.2.1.5 Automatically Registering Top and Side Faces as Machining Surfaces

By pressing the menu [Profile auto set], the top, bottom and four side faces of the workpiece are automatically registered as machining surfaces. The coordinate systems (feature coordinate systems) of each surface are also registered automatically.

Operation method

(1) Select the menu [Profile auto set]. → The menu will be highlighted and the confirmation message "Execute auto-registration?(Y/N)" is displayed.

(2) Press the [Y] or [INPUT] key. → The automatic registration operates and the message "Auto-registration completed" is displayed.

Note

(1) This operation is valid when the workpiece form is block. If the form is other than block, the message "Setting error" is displayed.

(2) When any of the workpiece size (X, Y and Z directions) is not set, the menu [Profile auto set] will be grayed out and non-selectable.

(3) The surface names to be registered are as follows:

TOP-SURFACE	A surface whose normal direction is +Z of the basic coordinate system
UNDER-SURFACE	A surface whose normal direction is -Z of the basic coordinate system
SIDE-SURFACE-1	A surface whose normal direction is +X of the basic coordinate system
SIDE-SURFACE-2	The side surface that comes first when turning clockwise about Z axis from SIDE-SURFACE-1
SIDE-SURFACE-3	The side surface that comes second when turning clockwise about Z axis from SIDESURFACE-1
SIDE-SURFACE-4	The side surface that comes third when turning clockwise about Z axis from SIDE-SURFACE-1

(4) This operation is disabled if 11 or more surfaces are registered besides those registered by this operation. The message "Setting error" is displayed.

(5) If no surface name is registered, but the surface's coordinate system has been registered (the origin and coordinate axis directions of the feature coordinate system are other than zero), the surface is recognized as registered.

(6) When the workpiece size is changed, this operation needs to be performed again because the coordinate systems set to each surface will shift. The setting of surface registered by this operation will be overwritten. However, the surface that is not able to overwrite (the registered surface is deleted or the name is changed) the setting will be registered again. (A renamed surface will be treated as the other surface.)

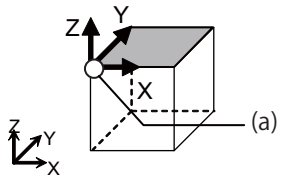
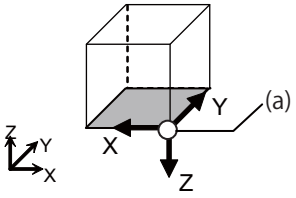
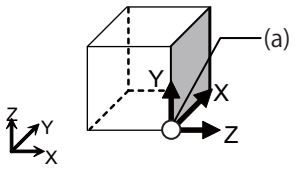
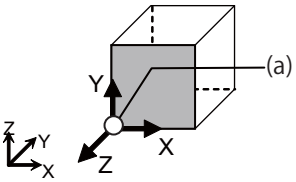
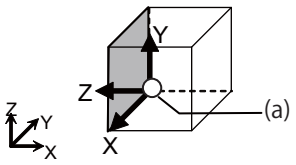
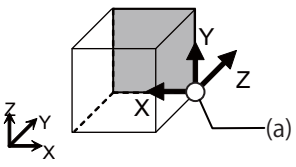
(7) If a surface to register by this operation is the machining target surface (a surface along which three orthogonal coordinate axes are controlled at present), this operation cannot be operated. The message "Can't set machining surface" is displayed.

(8) When the data protection is valid, this operation will be disabled. The message "Data protect" is displayed.

(9) If there are two surfaces having the same name to be overwritten, the surface with the smaller No. will be overwritten. If the surface is the machining target surface this operation cannot be operated. The message "Can't set machining surface" is displayed.

(10) There will be no data to be changed when error occurred and automatic register fails.

(11) Listed below are the surface names and data that are automatically set with [Profile auto set].
These are automatically calculated from the workpiece settings so that (a) is the origin.

Surface name	Feature coordinate origin	Axis direction	Method	Setting value
TOP-SURFACE		Z axis direction	A point on the axis (+)	X: 0.000 Y: 0.000 Z: 1.000
		X axis direction	A point on the axis (+)	X: 1.000 Y: 0.000 Z: 0.000
UNDER-SURFACE		Z axis direction	A point on the axis (+)	X: 0.000 Y: 0.000 Z: -1.000
		X axis direction	A point on the axis (+)	X: -1.000 Y: 0.000 Z: 0.000
SIDE-SURFACE-1		Z axis direction	A point on the axis (+)	X: 1.000 Y: 0.000 Z: 0.000
		X axis direction	A point on the axis (+)	X: 0.000 Y: 1.000 Z: 0.000
SIDE-SURFACE-2		Z axis direction	A point on the axis (+)	X: 0.000 Y: -1.000 Z: 0.000
		X axis direction	A point on the axis (+)	X: 1.000 Y: 0.000 Z: 0.000
SIDE-SURFACE-3		Z axis direction	A point on the axis (+)	X: -1.000 Y: 0.000 Z: 0.000
		X axis direction	A point on the axis (+)	X: 0.000 Y: -1.000 Z: 0.000
SIDE-SURFACE-4		Z axis direction	A point on the axis (+)	X: 0.000 Y: 1.000 Z: 0.000
		X axis direction	A point on the axis (+)	X: -1.000 Y: 0.000 Z: 0.000

TOP-SURFACE is a surface whose normal direction is +Z of the basic coordinate system.

UNDER-SURFACE is a surface whose normal direction is -Z of the basic coordinate system.

SIDE-SURFACE-1 is a surface whose normal direction is +X of the basic coordinate system.

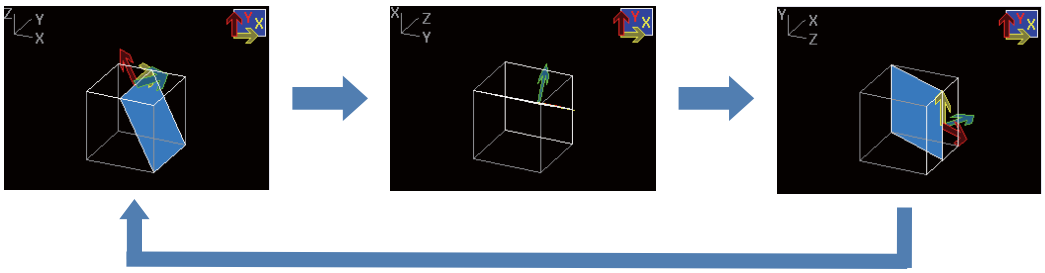
SIDE-SURFACE-2 to 4 are the surfaces obtained by rotating SIDE-SURFACE1 about the Z axis of the basic coordinate system.

9.2.1.6 Switching the Coordinate System Direction of a Workpiece Graphic

The coordinate system direction of the workpiece graphic (graphic display of the surface) can be switched to three coordinate system directions using the menu [Graphic dir] or [Graphic Rot]. The coordinate system direction of the workpiece graphic is reflected on the workpiece setting screen, Surface detail screen, and surface detail screen of the monitor screen, and the parameter "#8980 R-Navi graphic dir" is also updated.

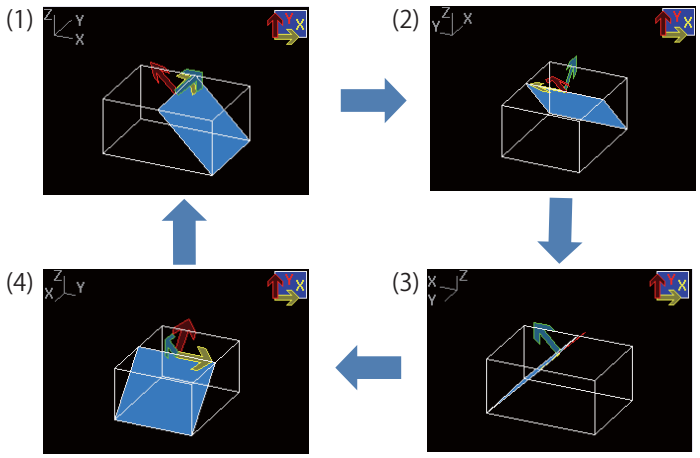
Each time the menu [Graphic dir] is pressed, the direction of the workpiece graphic is switched in the order of (1) to (3) as shown below:

(1) #8980 = 0 (XYZ) (2) #8980 = 1 (YZX) (3) #8980 = 2 (ZXY)



Each time the menu [Graphic Rot] is pressed, the direction of the workpiece graphic is switched in the order of (1) to (4) as shown below.

When "#8980 R-Navi graphic dir" = 0 (XYZ)



(1) Default view
(2) Rotated by 90 degrees from the default view
(3) Rotated by 180 degrees from the default view
(4) Rotated by 270 degrees from the default view

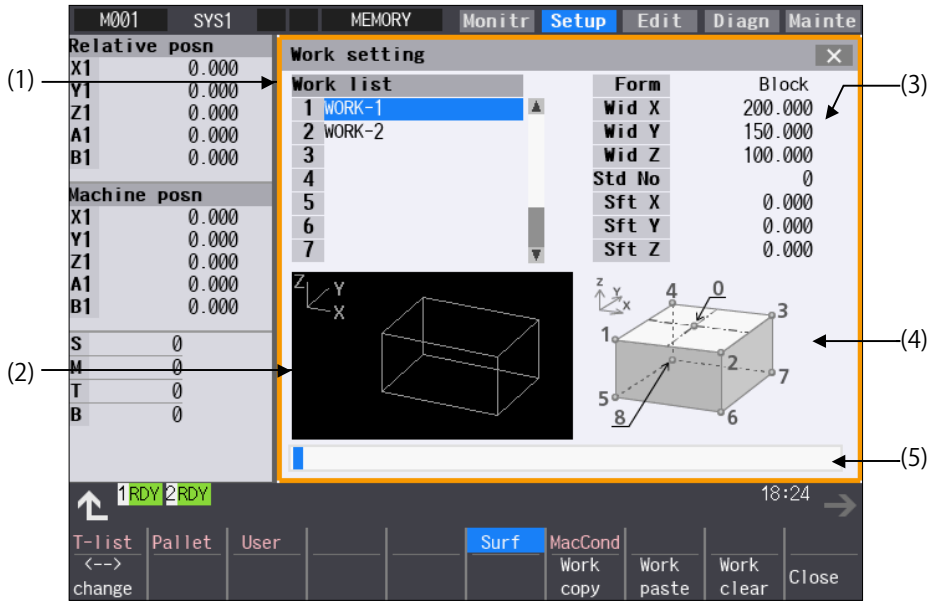
Note

(1) When data protection is valid, the coordinate system direction of the workpiece graphic can be switched using the menu, but the parameter value is not rewritten. In this case, when the power is turned ON again, the coordinate system direction of the workpiece graphic reverts to the same graphic direction that the parameter value is in.

(2) The view direction of the guide drawing is not switched due to the setting of the parameter or this menu.

9.2.2 Registering and Setting the Workpiece (on the Work Setting Screen)

When the menu [Work setting] is pressed the workpiece setting screen opens as a pop-up window. On this screen, name, form, size, and position of the workpiece can be set.



Display items

	Display items	Description
(1)	Work list	Displays a list of currently registered workpieces. Up to 10 workpieces can be registered in this list.
(2)	Workpiece graphic	Displays the form of the workpiece in 3D which is pointed by the cursor in the list.
(3)	Work setting	Displays the setting value of the workpiece pointed by the cursor in the list. The number of digits after the decimal point will obey the parameter #1003 iunit of the 1st part system. (Example) - 99999.999 to 99999.999 (mm) (When "#1003 iunit"= B)
(4)	Guide graphic	Displays the guide graphic that is related to the workpiece pointed by the cursor in the list.
(5)	Input section	Displays the key entered. The data entered here will be reflected in the current cursor position when the [INPUT] key is pressed.

Menus when the cursor is on the workpiece list

Menus	Description
<--> change	This moves the cursor between workpiece list and workpiece setting.
Work copy	This copies the data of workpiece pointed by the cursor (by one line). (*1)(*3)(*4)(*5)
Work paste	This writes the copied workpiece data (one line) to the line where the cursor is displaying. (*1)(*2)(*3)(*5) The copied workpiece data will be deleted when pasted.
Work clear	This clears the workpiece data of the specified line (multiple lines possible). (*5)(*6) Methods: Specify the No. of workpiece data to start and the No. to finish deleting. (Example) 1/E: Delete all workpiece data. The line where cursor is displaying will be deleted if the [INPUT] key is selected without specifying the line.
Close	This closes the pop-up window and quits this function.

(*1) If the surface data being copied is changed, the data will be revised when it is pasted.

(*2) This operation cannot be operated to the workpiece that includes the machining target surface (a surface along which three orthogonal coordinate axes are controlled at present). The message "Can't paste work including mach surface" is displayed.

(*3) If the surface data being copied is deleted, the paste cannot be done. The message "Paste error" is displayed.

(*4) The copied data will be deleted when the window is closed.

(*5) When the data protection is valid, this operation will be disabled. The message "Data protect" is displayed.

(*6) When a line is deleted, this operation is disabled for a workpiece that includes the machining target surface (a surface along which three orthogonal coordinate axes are controlled at present) and the message "Can't clear machining surface" is displayed.
If the range specified for this operation includes the workpiece that has the machining target surface (a surface along which three orthogonal coordinate axes are controlled at present) (1/E, for example), the workpieces except for the workpiece that includes the machining target surface will be deleted.

Menus when the cursor is on the workpiece setting list

Menus	Description
<--> change	This moves the cursor between workpiece list and workpiece setting.
Block	The workpiece form will turn to block when this menu is selected. If the cursor is outside the form area, this menu will be grayed out and non-selectable.
Pillar	The workpiece form will turn to pillar when this menu is selected. If the cursor is outside the form area, this menu will be grayed out and non-selectable.
Close	This closes the pop-up window and quits this function.

9.2.2.1 Registering the Workpiece

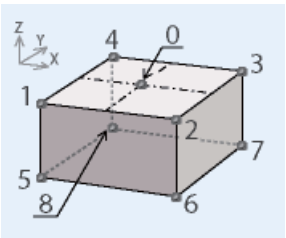
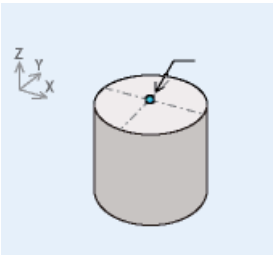
Operation method

(1) Move the cursor to the workpiece list by using the menu [<--> change].
Also, the cursor can be moved by tab keys ([|←], [→|]) to workpiece list.
➡ Move the cursor to the top of the workpiece list.

(2) Move the cursor to a line where no workpiece is registered.

(3) Input the workpiece name.
(If naming the workpiece is unnecessary, select [INPUT].)
➡ The name of workpiece is registered, and the cursor moves to the form line of workpiece setting.
Also, the menu corresponding to the value of form is valid.

(4) Select the form menu ([Block], [Pillar]) that needs to be set.
Setting contents differs by the selected workpiece form.
➡ The selected form will be reflected to the line of form and the cursor moves down a line.

Form	Guide graphic	Item	Description	Setting range (*1)
Block		Wid X	Width in the X direction	0.001mm(*2) - 99999.999mm
		Wid Y	Width in the Y direction	0.001mm(*2) - 99999.999mm
		Wid Z	Width in the Z direction	0.001mm(*2) - 99999.999mm
		Std No	Set a point to serve as a reference when setting the basic coordinate system's origin by the No. on guide graphic.	0: A center of the top 1 to 4: An angle of the top 5 to 8: An angle of the bottom
		Sft X	Shift distance from the point of Std No to the basic coordinate system's origin (in the X direction)	-99999.999mm - 99999.999mm
		Sft Y	Shift distance from the point of Std No to the basic coordinate system's origin (in the Y direction)	-99999.999mm - 99999.999mm
		Sft Z	Shift distance from the point of Std No to the basic coordinate system's origin (in the Z direction)	-99999.999mm - 99999.999mm
Pillar		Dia D	Diameter of the pillar	0.001mm(*2) - 99999.999mm
		Hgt Z	Height of the pillar	0.001mm(*2) - 99999.999mm
		Sft X	Shift distance from the top face's center to the basic coordinate system's origin (in the X direction)	-99999.999mm - 99999.999mm
		Sft Y	Shift distance from the top face's center to the basic coordinate system's origin (in the Y direction)	-99999.999mm - 99999.999mm
		Sft Z	Shift distance from the top face's center to the basic coordinate system's origin (in the Z direction)	-99999.999mm - 99999.999mm

(*1) These values follow the setting of "#1003 iunit" for the 1st part system.

(*2) Inputting 0 is possible.

(5) Input the workpiece size. → The value is input to the line where cursor is displaying and the cursor moves down a line.

Note

(1) Despite the presence of workpiece name, the workpiece will be treated as registered when its size meets the followings:

Block: The three axes (X, Y and Z) are defined
Pillar: The diameter D and height Z are defined

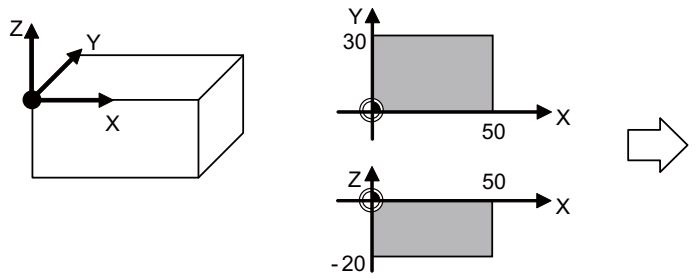
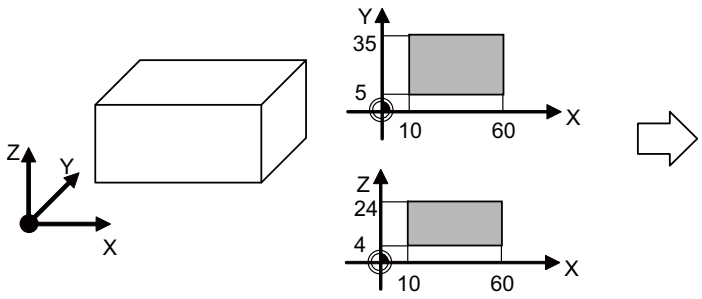
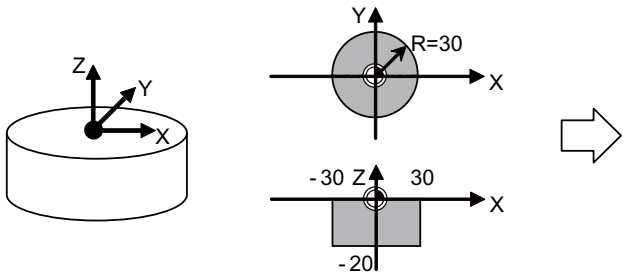
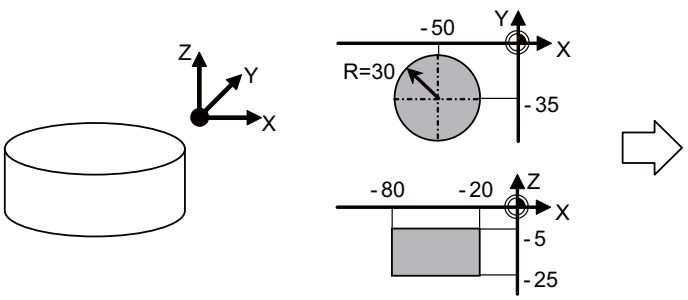
(2) Set the workpiece name with up to 20 one-byte characters (numbers, upper-case alphabets and the symbols that the system recognizes). If a one-byte lower-case alphabet is used for naming, it will be changed to the upper-case letter. Note that the following characters cannot be used:

\ / : , * ? " < > | space

A character string made of a single-byte zero only cannot be registered as a work name.

(3) When the data protection is valid, registering a workpiece is disabled. The message "Data protect" is displayed.

Work size setting examples

	Form: Block Wid X: 50.000 Wid Y: 30.000 Wid Z: 20.000 Std No: 1 Sft X: 0.000 Sft Y: 0.000 Sft Z: 0.000
	Form: Block Wid X: 50.000 Wid Y: 30.000 Wid Z: 20.000 Std No: 5 Sft X: -10.000 Sft Y: -5.000 Sft Z: -4.000
	Form: Pillar Dia D: 60.000 Hgt Z: 20.000 Sft X: 0.000 Sft Y: 0.000 Sft Z: 0.000
	Form: Pillar Dia D: 60.000 Hgt Z: 20.000 Sft X: 50.000 Sft Y: 35.000 Sft Z: 5.000

9.2.2.2 Changing the Workpiece Name

Operation method

(1) Move the cursor to the workpiece list by using the menu [<--> change].
Also, the cursor can be moved by tab keys ([|←], [→|]) to workpiece list.
➡ Move the cursor to the top of the workpiece list.

(2) Move the cursor to a line where the workpiece is registered using arrow keys ([↑], [↓]).

(3) Input the workpiece name.
To clear the workpiece name, input "0"(one-byte 0).
➡ The name of workpiece is changed, and the cursor moves to the form line of workpiece setting.

Note

(1) Form, size, data of the surface will not change by changing the name.

(2) The name of workpiece for which the machining target surface (a surface along which three orthogonal coordinate axes are controlled at present) has been registered, cannot be changed.
The message "Can't set work including mach surface" is displayed.

(3) When the data protection is valid, a workpiece name cannot be changed. The message "Data protect" is displayed.

9.2.2.3 Changing the Form of the Work

Operation method

(1) Move the cursor to workpiece setting by using the menu [<--> change].
If the cursor is already on the workpiece setting, move the cursor to the line of the workpiece shape by using arrow keys ([↑], [↓]).
The cursor can be moved by tab keys ([|←], [→|]) to workpiece setting.
Also, the cursor will move to workpiece setting by selecting the [INPUT] key when the cursor is in the workpiece list.
➡ Move the cursor to the top of workpiece setting.
The menu corresponding to the value of form becomes valid.

(2) Select the form menu ([Block], [Pillar]) that needs to be set.
➡ The selected form will be reflected to the line of form and the cursor moves down a line.
The guide graphic changes according to the selected form.

Note

(1) This operation cannot be operated to the workpiece that includes the machining target surface (a surface along which three orthogonal coordinate axes are controlled at present). The message "Can't set work including mach surface" is displayed.

(2) Even if the form changes, the size to be displayed is the data before the change, and the data of surface will not be changed.

(3) When the data protection is valid, a workpiece form cannot be changed. The message "Data protect" is displayed.

9.2.2.4 Changing the Workpiece Size

Operation method

(1) Move the cursor to workpiece setting by using the menu [<--> change]. If the cursor is already on the workpiece setting this operation is unnecessary.
The cursor can be moved by tab keys ([|←], [→|]) to workpiece setting.
Also, the cursor will move to workpiece setting by selecting the [INPUT] key when the cursor is in the workpiece list.
→ Move the cursor to the top of workpiece setting.

(2) Move the cursor to the line of size that needs to be set using arrow keys ([↑], [↓]).

(3) Input the workpiece size.
→ The value is input to the line where cursor is displaying and the cursor moves down a line.

Note

(1) This operation cannot be operated to the workpiece that includes the machining target surface (a surface along which three orthogonal coordinate axes are controlled at present). The message "Can't set work including mach surface" is displayed.

(2) The data of surface will not change by changing the size. Set again on the surface detail screen.

(3) When the data protection is valid, a workpiece size cannot be changed. The message "Data protect" is displayed.

(4) The setting contents differ by selected workpiece form. Refer to "9.2.2.1 Registering the Workpiece" for details.

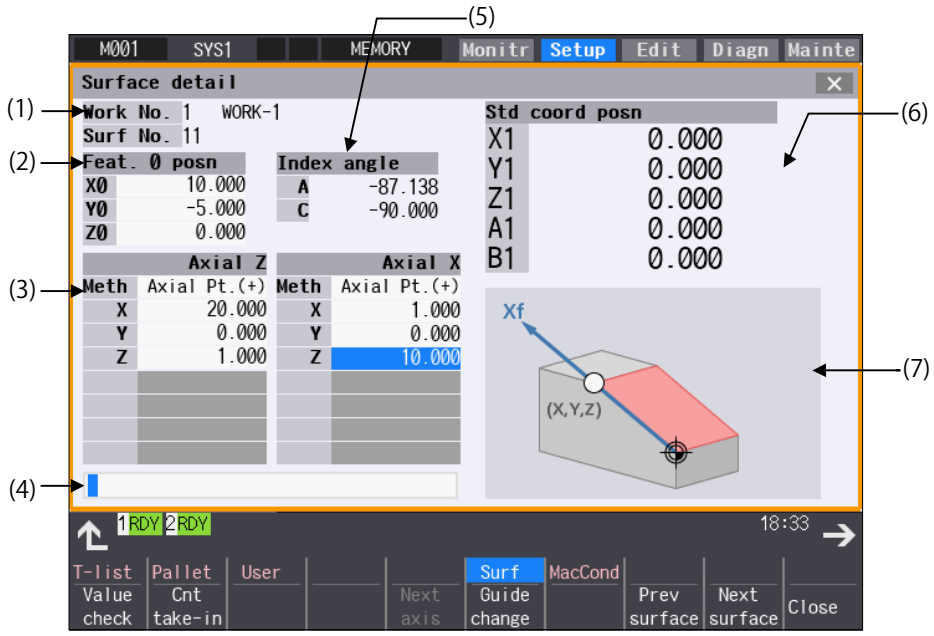
9.2.3 Setting the Details of the Surface (on the Surface Detail Screen)

By selecting the menu [Detail set] of the surface list screen, this opens the surface detail screen as a pop-up window. On this screen the coordinate origin (feature coordinate origin) and coordinate axis orientation of a coordinate system defined along the surface can be set.

Note

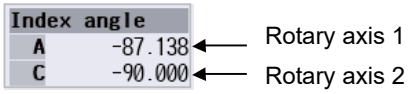
(1) This screen is not displayed in the following case. The message "Setting error" is displayed.

♦When the surface "BASE-SURFACE" is selected on the surface list screen.

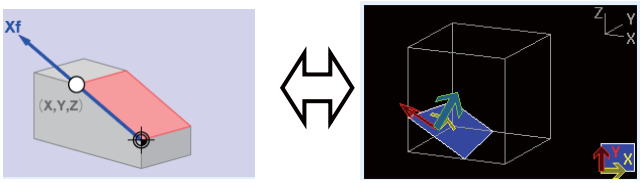


Display items

Display items		Description
(1)	Workpiece No. / Surface No.	Displays the work and surface of current settings.
(2)	Feature coordinate origin	Displays the settings of surface coordinate origin (feature coordinate origin). The number of digits after the decimal point will obey the parameter #1003 iunit of the 1st part system. (Example) - 99999.999 to 99999.999 (mm) (When "#1003 iunit" = B)
(3)	Axis direction setting	Displays the designation type of two coordinate axis directions of the surface, and the values (coordinates, rotation type, etc.) needed for the designation type. The initial values of the designation type and the axis combination can be set by #8954 and #8955. The number of digits after the decimal point will obey the parameter #1003 iunit of the 1st part system. (Example) - 99999.999 to 99999.999 (mm) (When "#1003 iunit" = B)
(4)	Input section	Displays the key entered. The data entered here will be reflected in the current cursor position when the [INPUT] key is pressed.

Display items		Description
(5)	Index angle	Displays the index angle calculated from the values set to the currently displaying two coordinate axes. The number of digits after the decimal point is fixed to three. The axis names to be displayed on the index angle are different according to the machine configuration.  •Tool tilt type Displays the axis name of #7932 ROTAXT2 for "Rotary axis1", the axis name of #7922 ROTAXT1 for "Rotary axis 2". •Table tilt type Displays the axis name #7942 ROTAXW1 for "Rotary axis1", the axis name #7952 ROTAXW2 for "Rotary axis 2" •Combined type Displays the axis name #7932 ROTAXT2 for "Rotary axis1", the axis name #7952 ROTAXW2 for "Rotary axis 2"
(6)	Basic coordinate position counter	Displays the value of the basic coordinate position counter (the counter that shows the coordinate position of basic coordinate system). Use the menu [Next axis] to open the counter of the 6th or subsequent axis.
(7)	Guide graphic/workpiece graphic	Displays either the workpiece graphic or the guide graphic that illustrates the registration method of the currently selected coordinate axes. Switching between guide graphic and work graphic is performed by using the menu [Guide change].

Menus

Menus	Description
Value check	This menu enables you to check if it is possible to define a surface using the set coordinate axial directions.
Cnt take-in	This sets the cursor position values in the basic coordinate position counter. <Note> ♦The menu will be grayed out if the cursor is in display item (3) "Axis direction setting" and the counter take-in has been disabled by the selected designation type.
Next axis	This switches the axes displayed on the basic coordinate position counter between the 1st to 5th axes and the 6th and subsequent axes when the valid axis number is six or more. <Note> ♦The menu will be grayed out and non-selectable when the number of valid axes is five or less.
Guide change	This switches between guide graphic and work graphic. [Guide graphic] [Work graphic] 
Prev surface	This switches the setting surface to the previous surface. If there is no previous surface, the last surface is displayed.
Next surface	This switches the setting surface to next surface. If there is no next surface, the first surface is displayed.
Close	This closes the pop-up window and quits this function.
Comb change	This changes the combination of two coordinate axes to be specified.
Axial rotate	This menu rotates the coordinate system (reversing the Z axis direction or rotating X/Y coordinates by 90 deg.) with respect to the coordinate axes defined along the machining surface.

Note

(1) The menus on the 1st page are different if the cursor is on the axial direction setting type.

9.2.3.1 Moving the Cursor

The cursor moves by the tab keys ([|←], [→|]) and the arrow keys ([↑], [↓]) on the surface detail screen.

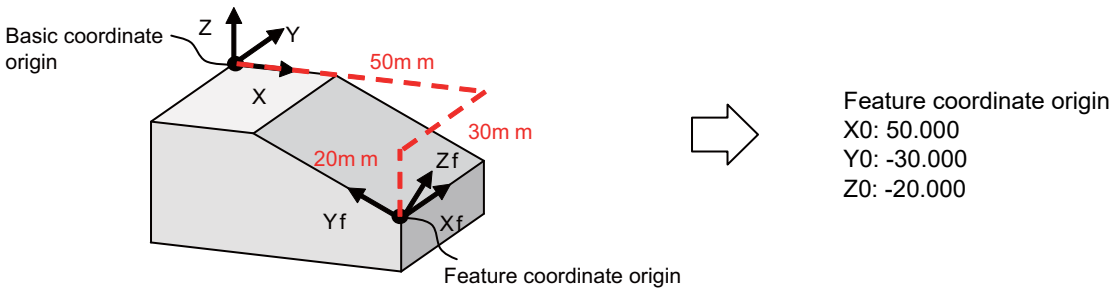
9.2.3.2 Setting the Surface Coordinate Origin (Feature Coordinate Origin)

Specify the surface coordinate origin (feature coordinate origin). The value of the coordinate origin is the offset from the basic coordinate system's origin. Also the X, Y and Z directions are those of the basic coordinate system.

Note

(1) When this operation is performed for the machining target surface (a surface along which three orthogonal coordinate axes are controlled at present), the error message "Can't set machining surface" is displayed.

(2) When the data protection is valid, entering or changing data is disabled on this screen. The message "Data protect" is displayed.



Operation method

(1) Move the cursor to the feature coordinate origin. → The cursor moves to the top of the feature coordinate origin.

(2) Move the cursor to the line to set the value of origin.

<When inputting the value directly>

(3) Input the value of origin. → The inputted value will be set in the cursor position.

<To take in the value of the basic coordinate position counter (a counter that displays the position with respect to the basic coordinate system)>

(3) Select the menu [Cnt take-in]. → The value of the basic coordinate position counter will be set to the position of the counter.

Note

(1) The counter values of the axes specified by the parameters #7900 RCDAX_I to #7902 RCDAX_K will be taken. The relation between the feature coordinate origin and the parameters is listed below. If any of the parameter settings is illegal or any of the axes is set as non-display ("#1069 no_dsp" is "1"), the counter take-in will be disabled.
In this case the message "Counter take in not possible" will appear.

Feature coordinate origin	Corresponding parameter
X0	#7900 RCDAX_I
Y0	#7901 RCDAX_J
Z0	#7902 RCDAX_K

9.2.3.3 Specifying the Surface Coordinate Axial Directions

Specify the surface coordinate axial directions. There are five types of methods for specifying the coordinate axial directions: "Axial point (+)", "Longitude/Latitude", "Longitude/Projection angle", "Start point/End point" and "Index angle". "Axial point (+)", "Start point/End point", and "Index angle" can take in the value of the basic coordinate position counter by selecting the menu [Cnt take-in].

Note

(1) When this operation is performed for the machining target surface (a surface along which three orthogonal coordinate axes are controlled at present), the error message "Can't set machining surface" is displayed.

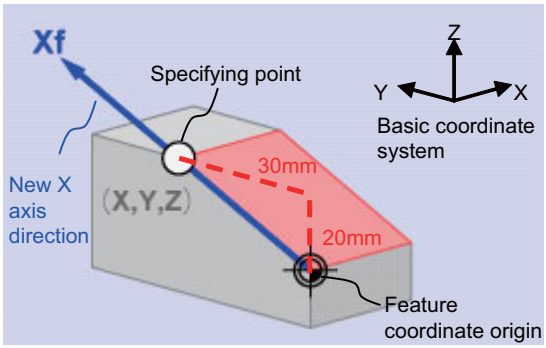
(2) When the data protection is valid, entering or changing data is disabled on this screen. The message "Data protect" is displayed.

[Method 1] Axial point (+)

With this method you specify one point, except for the feature coordinate origin, using an offset from the feature coordinate origin (an offset in the basic coordinate system direction), and the coordinate axial direction is determined by this point and the feature coordinate origin. The direction from the feature coordinate origin to the designated point is positive (+).

[Setting value]

Offset from the coordinate origin (feature coordinate origin).



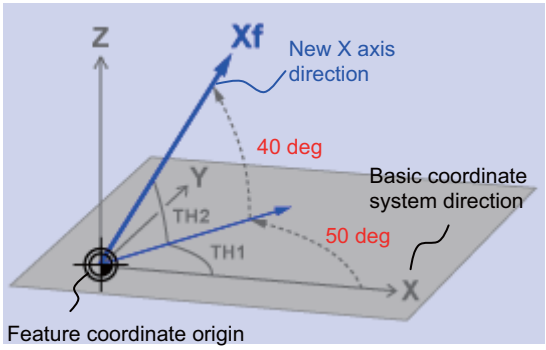
X axis direction
X: 0.000
Y: 30.000
Z: 20.000

[Method 2] Longitude/Latitude

This method specifies coordinate axial direction by rotating the axes of the basic coordinate system by the designated longitude and latitude. The plane to be used as the basis of the longitude and latitude varies depending on which coordinate axis to set.

[Setting value]

Coordinate axis	θ1 (TH1)	θ2 (TH2)
X axis direction	Rotation angle about Z axis	Angle from XY plane
Y axis direction	Rotation angle about Z axis	Angle from XY plane
Z axis direction	Rotation angle about X axis	Angle from YZ plane



X axis direction
TH1: 50.000
TH2: -40.000

[Method 3] Longitude/Projection angle

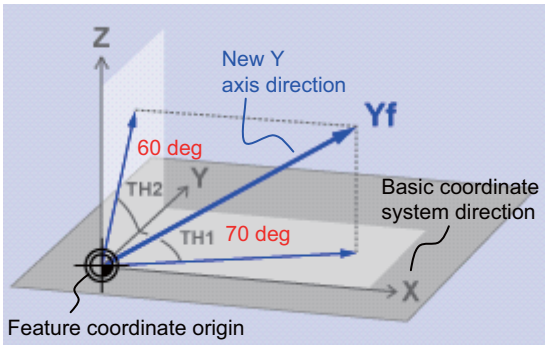
This method specifies coordinate axial direction by rotating the axes of the basic coordinate system by the designated longitude and latitude. The plane to be used as the basis of the longitude and latitude varies depending on which coordinate axis to set.

[Setting value]

Coordinate axis	θ1 (TH1)	θ2 (TH2)
X axis direction	Rotation angle about Z axis	Project angle on ZY plane
Y axis direction	Rotation angle about Z axis	Project angle on YZ plane
Z axis direction	Rotation angle about X axis	Project angle on ZY plane

Note

(1) If you set 90 deg. or -90 deg. for TH1, the projected angle on the plane cannot be calculated. In this case the directions after rotated by TH1 will be the coordinate axial directions, regardless of the TH2 value.



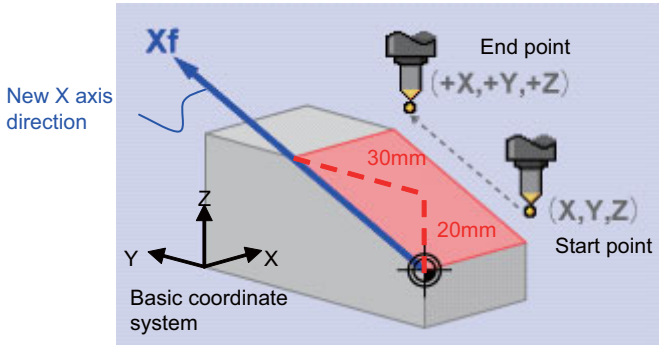
Y axis direction
TH1: -70.000
TH2: 60.000

[Method 4] Start point/End point

With this method you specify the start- and end-point coordinates by manual operation to specify the coordinate axial direction in the same direction as from the specified start-point to the end-point. The coordinate origin is that of the feature coordinate system, and the direction from the start point to the end point is positive.

[Setting value]

X, Y, Z: Start-point coordinate (basic coordinate position)
+X, +Y, +Z: End-point coordinate (basic coordinate position)



X axis direction
(Start point)
X : 30.000
Y : -50.000
Z : 25.000
(End point)
+X : 30.000
+Y : -20.000
+Z : 45.000

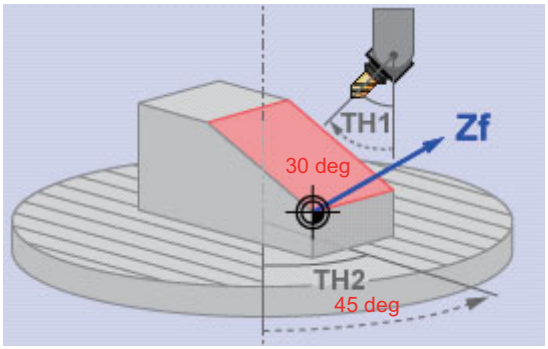
[Method 5] Index angle

With this method you directly specify the indexing angle in the Z axis direction. This method is unusable for the X and Y directions. The message "Setting error" is displayed. If you select the "Index angle" method for Z axis , and select the [Axial point (+)] or [Start point/End point] method for X and Y axes with all the settings set to "0.000", the X/Y directions are controlled to be in the basic coordinate system directions after surface indexing, regardless of the methods for X and Y. (When the X/Y directions have been set, the setting values will be observed.)

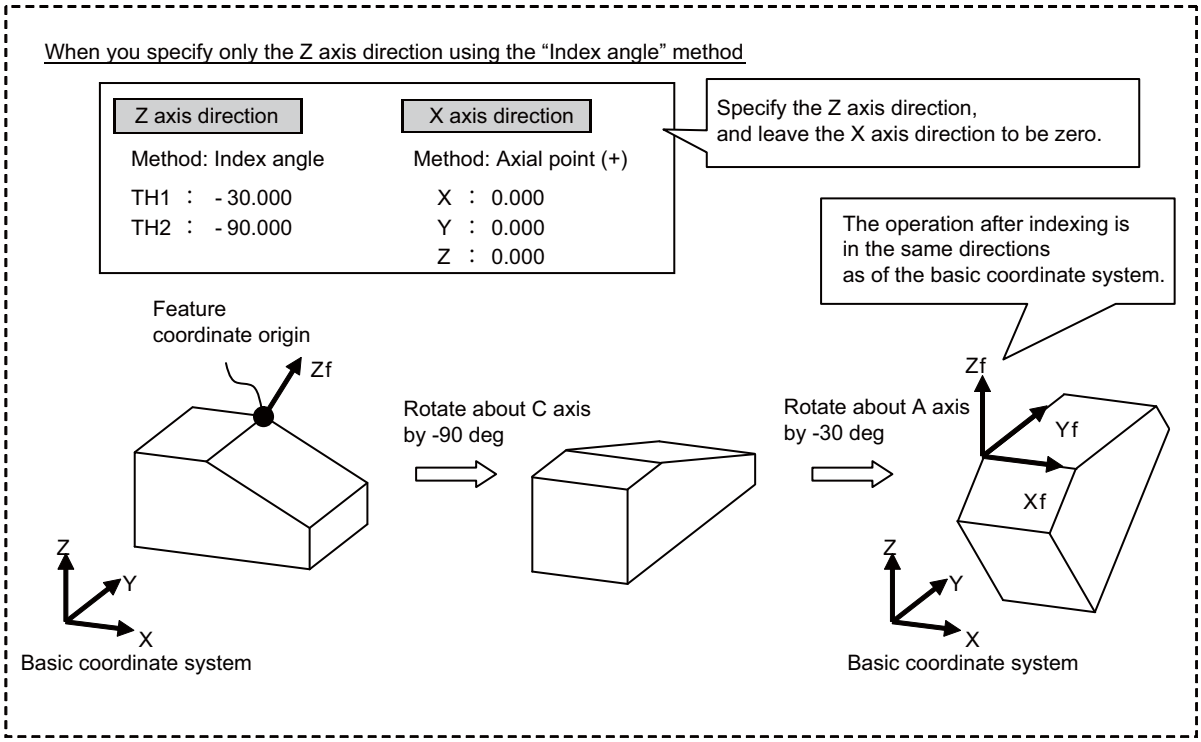
[Setting value]

θ1(TH1): The angle of rotary axis B
θ2(TH2): The angle of rotary axis C

(When the rotary axis is set to B or C)



Z axis direction
TH1: 30.000
TH2: -45.000



Menus

When the cursor is moved to "Meth" (axial direction designation type), the menus on the 1st page will switch to the following ones. (The menus on the 2nd page will not change.)

Menus	Description
Axial Pt.(+)	This is the method to specify the coordinate axial direction using an optional point (except for feature coordinate origin) and feature coordinate origin.
Long./ lat.	This method specifies coordinate axial direction by rotating the axes of the basic coordinate system by the designated longitude and latitude.
Long./ p-angle	This is the method to specify the coordinate axial direction by rotating the basic coordinate system axis by the specified longitude/projection angle.
StartPt /EndPt	This is the method to specify the coordinate axial direction using the manually specified start-point and end-point.
Index angle	With this method you directly specify the indexing angle in the Z axis direction.

Operation method

(1) Move the cursor to "Meth". → The menus change.

(2) Select the menu [Long./ lat.] → The value of the specified method will be longitude/ projection angle.

<When inputting the value directly>

(3) Input the value. → The inputted value will be set in the cursor position.

<To take in the value of the basic coordinate position counter (a counter that displays the position with respect to the basic coordinate system)>

(3) Select the menu [Cnt take-in]. → The value of the basic coordinate position counter will be set to the position of the counter.
After setting, the cursor moves down a line.

Note

(1) Which axis to use for taking in the counter value is determined by the parameters. The relation between the axes for taking in counter value and the parameters is listed below. If any of the parameter settings is illegal or any of the axes is set as non-display ("#1069 no_dsp" is "1"), the counter take-in will be disabled. In this case the message "Counter take in not possible" will appear.

When the designated type is Axial point (+)

Setting item	Corresponding parameter
X	#7900 RCDAX_I
Y	#7901 RCDAX_J
Z	#7902 RCDAX_K

When the designated type is Start point/End point

Setting item	Corresponding parameter
X	#7900 RCDAX_I
Y	#7901 RCDAX_J
Z	#7902 RCDAX_K
+X	#7900 RCDAX_I
+Y	#7901 RCDAX_J
+Z	#7902 RCDAX_K

When the designated type is Index angle

Setting item	Corresponding parameter		
	Tool tilt type	Table tilt type	Combined type
TH1	#7932 ROTAXT2	#7942 ROTAXW1	#7932 ROTAXT2
TH2	#7922 ROTAXT1	#7952 ROTAXW2	#7952 ROTAXW2

9.2.3.4 Checking the Setting Value of Coordinate Axis

Pressing the menu [Value check] enables you to check if it is possible to define a surface using the set coordinate axial directions. When defining a surface is allowed, the message "Setting value check completed" appears. In the following cases, defining a surface is judged as impossible, and the message "Can't define machining surface" appears.

(1) When the designation method is "axial point (+)" or the set value of "Start point/End point" is "0".
(When "Index angle" is selected for the Z axis, X/Y directions will be out of the scope of this check.)

(2) When an angle formed by the set two coordinate axes is either "0 deg. or greater but 1 deg. or smaller" or "179 deg. or greater but 180 deg. or smaller".

9.2.3.5 Changing the Combination of Axial Direction

You can change the coordinate axial direction combination (Z/X, Z/Y, X/Y) to be displayed on coordinate axial direction setting.

Operation method

(1) Select the menu [Comb change].	→	The menu switches. The menu of the currently selected axis combination will be highlighted.
(2) Select the menu [X/Y].	→	The coordinate axial direction of the coordinate axial direction setting changes to X/Y. The menu returns to the main menu of the surface detail screen.

Note

(1) The setting value of coordinate axis will not change even if the axis combination is changed. Please set it again.

(2) When this operation is performed for the machining target surface (a surface along which three orthogonal coordinate axes are controlled at present), the error message "Can't set machining surface" is displayed.

(3) When the data protection is valid, entering or changing data is disabled on this screen. The message "Data protect" is displayed.

9.2.3.6 Switching the Surface

Operation method

(1) Select the menus [Prev surface] and [Next surface]	→	<When the menu [Prev surface] is selected> This will change the surface to the previous surface. (This will also display the surfaces for which no coordinate system has been set.) If there is no previous surface, the last surface is displayed. The feature coordinate origin and coordinate axial direction of the surface after switch are displayed. The cursor will move to X0 of feature coordinate origin. <When the menu [Next surface] is selected> This will change the surface to the next surface. (This will also display the surfaces for which no coordinate system has been set.) If there is no next surface, the first surface is displayed. The feature coordinate origin and coordinate axial direction of the surface after switch is displayed. The cursor will move to X0 of feature coordinate origin.

9.2.3.7 Rotating the Axial Direction of Coordinate Axis

The coordinate axes of the coordinate system defined on a machining surface can be rotated.

- When this operation is performed for the machining target surface (a surface along which three orthogonal coordinate axes are controlled at present), the error message "Can't set machining surface" is displayed.
- When the data protection is valid, entering or changing data is disabled on this screen. The message "Data protect" is displayed.
- When "Index angle" method is applied to Z axis and "Axial point (+)" to X/Y axes, and the set values of "Star point/ End point" are all "0.000", it is not possible to rotate X and Y by 90 degrees.

Note

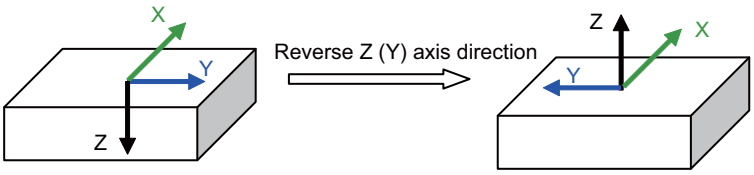
(1) When this operation is operated, the value of coordinate axial direction registered to NC and the value of coordinate axial direction displayed on the surface detail screen will be automatically changed.

(2) When you reverse the Z axis direction twice, or rotate X/Y axes by 90 degrees four times, you restore the original axial directions, but a small deviation may result due to a calculation error.

The following two types of operation are possible

Operation method (Reverse Z axis direction)

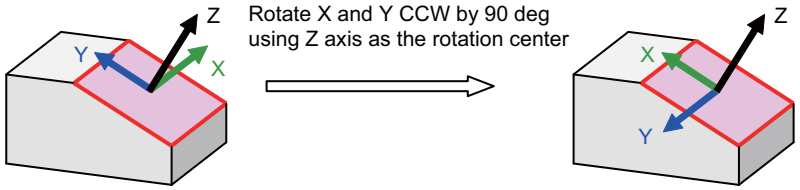
While fixing the X axis direction, inversely rotate the Z and Y axes by 180 deg.



(1) Select the menu [Axial rotate].

(2) Select the menu [Z-axis reverse]. ➡ The Z (Y) axis direction of the feature coordinate defined on the machining surface will reverse.

Operation method (Rotate X and Y CCW by 90 deg. (the center of rotation is Z axis))



(1) Select the menu [Axial rotate].

(2) Select the menu [X/Y 90 rotate]. ➡ The X and Y coordinates of the feature coordinate defined on the machining surface will rotate 90 deg. counter-clockwise about Z axis

9.2.3.8 Default Settings of Designation Method and Axis Combination

Use the following parameters to specify the default settings of the axis direction designation method and the two-axis combination to be used for designating the direction. Specify the initial setting of the designation method with the parameter "#8954 Initial type". Specify the initial setting of the axis combination with the parameter "#8955 Init axis pair".

When any of the following operations is carried out, the values of #8944 and #8955 will be set as default values.

Operation	The surface to which the parameter values are set
When the workpiece size is set for the first time	All surfaces belonging to the work (except for Surface #1 (BASE-SURFACE))
When the workpiece size is set again after the workpiece is cleared.	All surfaces belonging to the work (except for Surface #1 (BASE-SURFACE))
When the surface is cleared.	The cleared surface

Note

(1) When #8954 (Initial type) is set to 5 (Index angle), the "Index angle" method is applied to the Z axis direction. However, "Axial point (+)" is applied to the X and Y directions.

(2) If you change the parameter #8944 or #8955 after setting the workpiece size, the designation method or axis combination will not change. To reflect the value of the parameter, reset the work size after clearing the workpiece or clear the surface.

9.2.4 Surface Selection

Monitr – S-sel

By selecting a machining surface, the commands programmed with three orthogonal axes can move the tool along the selected surface (feature coordinate system). This selected surface is called a machining target surface. This enables a program with three orthogonal axes to carry out multi-surface machining including inclined surface machining. Also, a tool can be set vertical to the machining target surface (indexing).
This function is an optional specification, and the machining surface can be selected for each part system.

Note

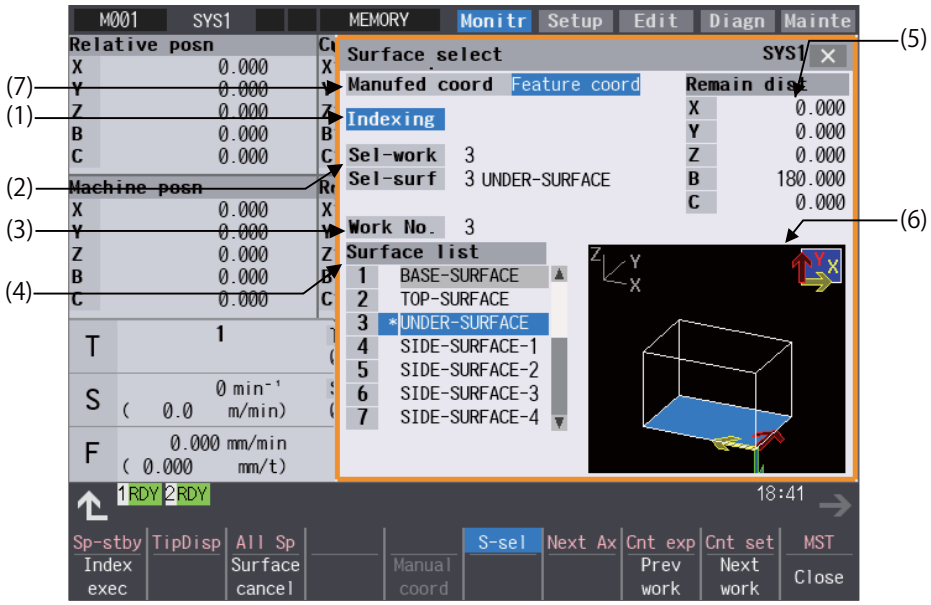
(1) To select a machining surface, work and machining surface need to be registered beforehand on the surface setting screen of setup screen. For details, refer to "9.2 R-Navi".

⚠ CAUTION

Selected machining surface is not canceled even when NC reset is executed, the coordinate system defined along the machining target surface (feature coordinate system) remains selected.

It depends on the setting of the parameter "#1247 set19/bit2" (Inclined surface machining mode hold) to hold or cancel the selected machining surface at the emergency stop state. However, selected machining surface is not held when the power is turned OFF.

Press the menu [Surface cancel] to cancel the selected machining surface. The operating coordinate system will be back to the machine coordinate system by canceling the machining surface.



Display items

	Display items	Description
(1)	Index state	Displays the index state. When "#11037 R-Navi Index Type" is "0", "Indexing" is displayed with flicker by selecting the menu [Index exec]. When "#11037 R-Navi Index Type" is "1", "Indexing" is displayed with flicker by selecting the menu [Index exec] and then entering the tool length offset number (H).
(2)	Selected work/Selected surface	Displays the machining target surface and its work of the part system being displayed.
(3)	Work No.	Displays the work No. that is displayed on the surface list. The work can be switched by selecting the menus [Prev work] and [Next work].
(4)	Surface list	Displays the list of surfaces that are registered for the work displayed at Work No. If the machining target surface is included, it is displayed with an asterisk (*).
(5)	Remaining distance counter	Displays the remaining distance up to the point where the tool is vertical to the machining target surface (indexing position). When "#11037 R-Navi Index Type" is "0", remaining distance will be calculated automatically by pressing the menu [Index exec]. When "#11037 R-Navi Index Type" is "1", remaining distance will be calculated automatically by pressing the menu [Index exec] and then inputting the tool length compensation number. The counters of five axes are displayed in the order specified by "#1494 dsp_ax_change".
(6)	Workpiece graphic	Displays the image of workpiece displayed at Work No. in 3D. It also displays with arrow marks, the surface pointed by the cursor in the list, and the directions of X, Y and Z axes of the coordinate system (feature coordinate system) registered for the surface. (X axis is in yellow, Y in red and Z in green)
(7)	Manual feed coordinate system	Displays the coordinate system along which manual feed is carried out while the machining target surface is being selected. The machine coordinate system and the feature coordinate system can be switched by the menu [Manual coord]. If the target surface is not selected it is displayed as blank. The initial coordinate values when a machining target surface is selected will be those of the feature coordinate system when the parameter "#8127 R-Navi Manufed coord" is "0" , and when #8127 is "1" the initial values will be those of the machine coordinate system. <Note> ♦The operation is performed along the feature coordinate system during automatic operation, regardless of the coordinate system displayed here.

Menus

Menus	Description
Index exec	Executes indexing to the machining target surface.
Surface cancel	Cancels the coordinate system (feature coordinate system) defined when a machining target surface is selected, and sets the current coordinate system as the machine coordinate system.
featcrd-shiftin	Displays the feature coordinate system shift screen.
Manual coord	Switches the coordinate system along which manual feed is performed during the machining target surface selection between the machine coordinate system and feature coordinate system. This menu is valid only during surface selection. (*1)
Prev work	Switches the work to display from the currently selected work to the previous work. If there is no previous workpiece, the last workpiece is displayed.
Next work	Switches the work to display from the currently selected work to the next work. If there is no next work, the first work is displayed.

Menus	Description
Close	This closes the pop-up window.

(*1) When the target surface is not selected, his menu is grayed out and cannot be selected.

9.2.4.1 Selecting a Machining Target Surface

Select a machining target surface. By selecting the machining surface, the commands programmed with three orthogonal axes can move the tool along the selected surface (feature coordinate system). This enables a program with three orthogonal axes to carry out multi-surface machining including inclined surface machining. Machining surface can be selected for each part system.

⚠ CAUTION

Selected machining surface is not canceled even when NC reset is executed, the coordinate system defined along the machining target surface (feature coordinate system) remains selected.

It depends on the setting of the parameter "#1247 set19/bit2" (Inclined surface machining mode hold) to hold or cancel the selected machining surface at the emergency stop state. However, selected machining surface is not held when the power is turned OFF.

Press the menu [Surface cancel] to cancel the selected machining surface. The operating coordinate system will be back to the machine coordinate system by canceling the machining surface.

Operation method

(1) Move the cursor to the row of machining surface to be selected using arrow keys ([↑], [↓]).

(2) Press the [INPUT] key.

➡ An asterisk (*) is placed in front of a surface name, indicating that the surface is the machining target surface. Also, the name of the selected work and its number, and the machining surface name and its number are displayed at Sel-work and Sel-surf.
If coordinate system setting of machining surface is illegal, it cannot be selected.
The message "Feat. coord setting error" is displayed.

Note

(1) The machining target surface and its work will be canceled when the power is turned ON again.

(2) This operation is disabled during automatic operation. The message "Executing automatic operation" is displayed.

(3) This operation is disabled during surface indexing. The message "Indexing machined surface" is displayed.

(4) This operation is disabled when rotary axis configuration parameter is illegal, surface selection is being processed (*1), or surface cancel is being processed (*2). The message "Machining surface cannot be selected" is displayed.

(*1) "Surface selection is being processed" means the period from when machining surface selection is performed (the [INPUT] key is pressed) until when the machining target surface is actually selected and the selected work and surface are displayed.

(*2) "Surface cancel is being processed" means the period from when machining surface cancel is performed (the menu [Surface cancel] is pressed) until the target surface is actually canceled and the display of the selected work and surface is canceled.

(5) On MDI status "SEL" is displayed with flicker when a machining surface is selected.
(Even under the MDI mode, "SEL" display is preferred during the surface selection.

(6) On the monitor screen of the 2-part system simultaneous display, "SEL" is not displayed with flicker on MDI status even if the part system that is under surface selection is displayed.

(7) The graphic check or trace is carried out along the machine coordinate system while a machining target surface is being selected.

9.2.4.2 Indexing the Machining Target Surface

Given below is the procedure of indexing the currently selected machining surface (turning the tool to be perpendicular to the machining target surface). By setting the parameter "#11037 R-Navi Index Type", method can be selected from two types: moving the rotary axis only (type 1) and moving both rotary and orthogonal axes while fixing the tool tip at a position viewed from the workpiece (type 2).

Operation method

[Prepare indexing]
<When #11037=0>

(1) Select the menu [Index exec]. → "Indexing" flickers and the machining target surface will be in the indexing state. The remaining distance necessary to complete indexing is displayed on the remaining distance counter.

<When #11037=1>

(1) Select the menu [Index exec]. → The menu [Index exec] will be highlighted, and the message "Set the tool length compensation No.(H)." is displayed.

(2) Input the tool length compensation No. and select the [INPUT] key. → "Indexing" flickers and the machining target surface will be in the indexing state. The remaining distance necessary to complete indexing is displayed on the remaining distance counter.

[Execute indexing]

<For manual indexing>
Use a handle to do indexing.

(1) Select the memory mode and select auto start button. → Operates automatic operation until the remaining distance counter becomes "0". Also, when the value of counter becomes "0", flickering "Indexing" will disappear.

<For manual indexing>
Use a handle to do indexing. This allows the operator to do indexing while observing the work and tool positions. It is also possible to switch to automatic indexing in the middle of the operation.

(1) Select the rotary axis in the handle mode and turn the handle. → When the counter values reach all zero, flickering "Indexing" will disappear.

Note

(1) This operation is disabled during automatic operation. The message "Executing automatic operation" is displayed.

(2) The indexing will be canceled by NC reset operation.

(3) This operation is disabled if no machining target surface is selected. The message "Machining surface not selected" is displayed.

(4) This operation is disabled during surface indexing. The message "Indexing machined surface" is displayed.

(5) This operation is disabled when surface selection is being processed (*1), or when surface cancel is being processed (*2). The message "Indexing not possible" is displayed.

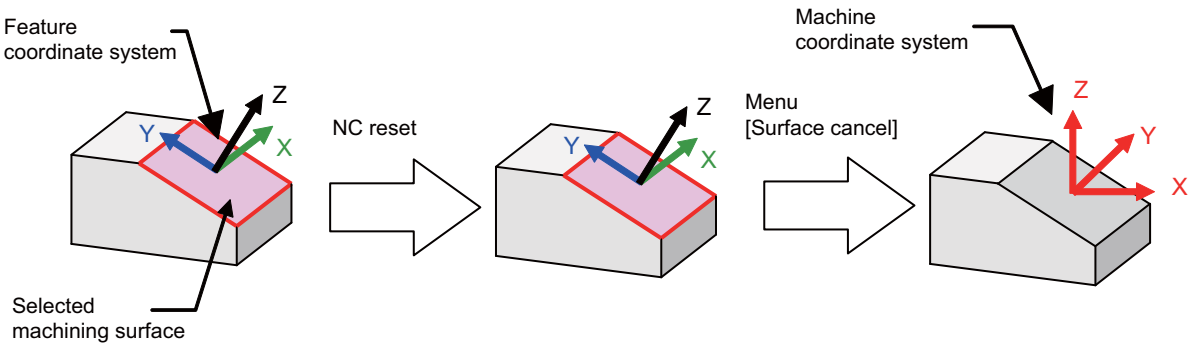
(*1) "Surface selection is being processed" means the period from when machining surface selection is performed (the [INPUT] key is pressed) until when the machining target surface is actually selected and the selected work and surface are displayed.

(*2) "Surface cancel is being processed" means the period from when machining surface cancel is performed (the menu [Surface cancel] is pressed) until the target surface is actually canceled and the display of the selected work and surface is canceled.

(6) When 3D machine interference check is valid, the manual indexing cannot be executed. The operation error (M01 0005) will occur if the indexing is executed.

9.2.4.3 Canceling the Machining Target Surface

When a machining surface is selected, the coordinate system defined on the machining surface (feature coordinate system) remains selected even if NC reset is executed. This operation, however, switches the selected coordinate system back to the machine coordinate system.



Operation method

(1) Select the menu [Surface cancel]. → The menu [Surface cancel] will be highlighted, and the message "OK?(Y/N)" is displayed.

To cancel the highlight, select the menu [Return] or select the menu [Surface cancel] again.

(2) Press the [Y] or [INPUT] key. → The selection of machining surface will be released.

Note

(1) The "Executing automatic operation" error occurs during automatic operation, and canceling the surface is disabled.

(2) This operation is disabled when surface selection is being processed (*1), or when surface cancel is being processed (*2). The message "Can't cancel" is displayed.

(*1) "Surface selection is being processed" means the period from when machining surface selection is performed (the [INPUT] key is pressed) until when the machining target surface is actually selected and the selected work and surface are displayed.

(*2) "Surface cancel is being processed" means the period from when machining surface cancel is performed (the [Surface cancel] key is pressed) until the target surface is actually canceled and the display of the selected work and surface is canceled.

(3) The flickering "SEL" on the MDI status will be canceled.

(4) This operation is disabled if no machining target surface is selected. The message "Machining surface not selected" is displayed.

(5) This operation is disabled during surface indexing. The message "Indexing machined surface" is displayed.

9.2.4.4 Switching the Machining Target Surface

When selecting the menu [Manual coord] during surface selection, the coordinate system along which manual feed is conducted can be switched between the machine coordinate system and feature coordinate system. The initial coordinate values when a machining target surface is selected will be those of the feature coordinate system when the parameter "#8127 R-Navi Manufed coord" is "0" , and when #8127 is "1" the initial values will be those of the machine coordinate system. However, operation is performed along the feature coordinate system during automatic operation, regardless of the coordinate system displayed here.

Operation method

Assume that the parameter "#8127 R-Navi Manufed coord" is set to "0" (feature coordinate system).

(1) Select a machining target surface. → The manual feed coordinate system becomes "feature coordinate system".

(2) Select the menu [Manual coord]. → The manual feed coordinate system becomes "machine coordinate system".

(3) Cancel the surface by the menu [Surface cancel]. → The manual feed coordinate system becomes blank.

(4) Select a machining target surface. → The manual feed coordinate system becomes "feature coordinate system".

9.2.4.5 Switching the Workpiece

Switch the work that is displayed on "Work No." to the other work. When work is switched, the cursor on the surface list will move to the surface "BASE-SURFACE".

Operation method (Select next work)

Switch the currently displayed work to the next work. If there is no next work, the first work is displayed.

(1) Select the menu [Next work]. → The work switches to the next largest No. of work.

Operation method(Select previous work)

Switch the currently displayed work to the previous work. If there is no previous workpiece, the last workpiece is displayed.

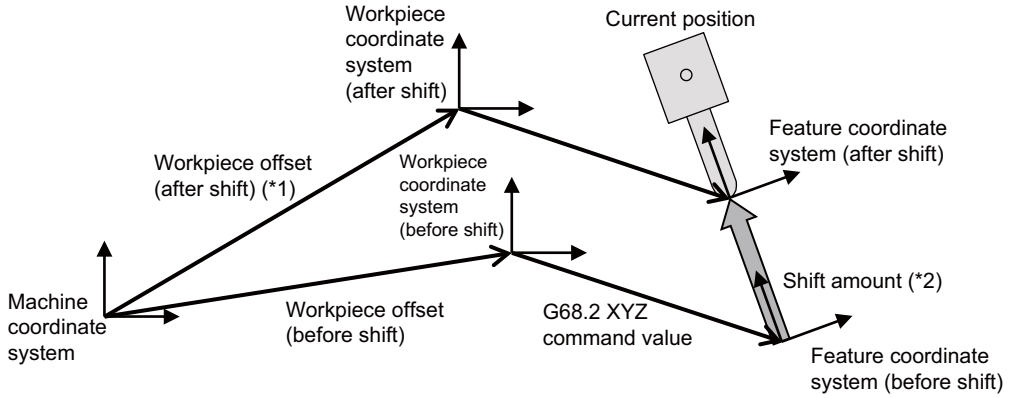
(1) Select the menu [Prev work]. → The work switches to the next smallest No. of work.

9.2.5 Shifting the Feature Coordinate Origin

The workpiece offset to shift the feature coordinate origin by the specified amount is automatically set. The shift amount is specified with a value on the feature coordinate system.

By using this function, the current position can be set to the feature coordinate origin or a position on the feature coordinate system.

[Example of feature coordinate system shift]



(*1) Automatically set according to the shift amount.
(*2) Specify the shift amount of the feature coordinate origin.

Note

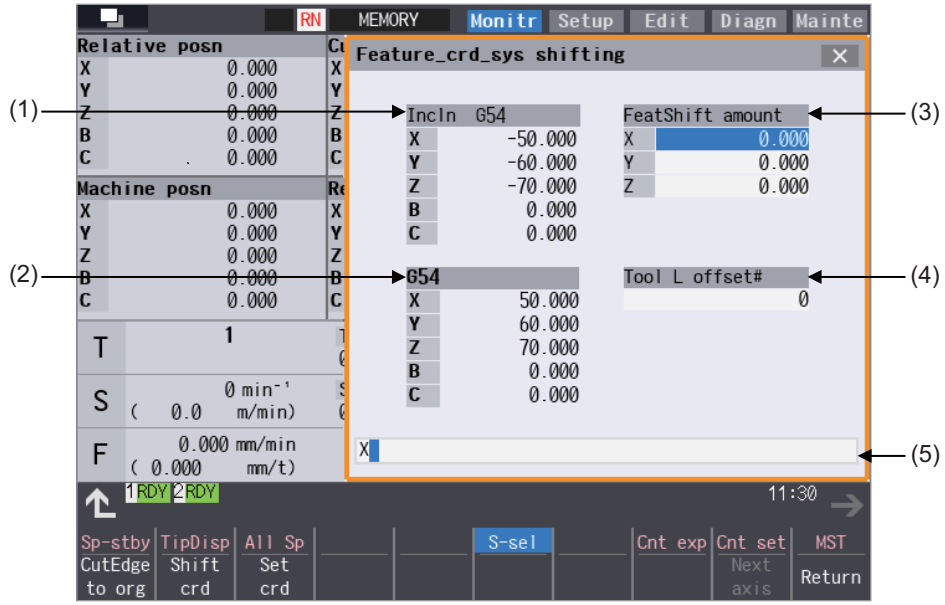
The feature coordinate system shift cannot be performed in the following cases.

State	Operation messages
In automatic operation (not during single block stop)	Executing automatic operation
Protected according to the operation level of data protection by user's level	Data protect
Data protection key 1 is enabled.	
During tool center point control	FeatCrdSysShifting NOT allowed
During tool cutting point control	
During workpiece installation error compensation	
In a mode where the inclined surface machining command cannot be issued (*1)	
Motion control release is valid.	

(*1) It indicates that the alarm "P952 Inclined face cut prohibited" occurs when the inclined surface machining (G68.2) is commanded.

By pressing the menu [featcrdshiftin] in the machining surface selection screen, the feature coordinate system shift screen is displayed.

When no machining surface is selected or while surface indexing is executed, the menu [featcrdshiftin] is grayed out and non-selectable.



Display items

Display items		Details
(1)	Inclined surface coordinate position	Displays the inclined surface coordinate position.
(2)	Workpiece coordinate system offset	Displays the workpiece offset of the basic coordinate system on the selected machining surface.
(3)	Feature coordinate system shift amount	Sets the movement amount to move the feature coordinate origin or the value of the moved inclined surface coordinate. These values follow the setting of "#1003 iunit" for the 1st part system.
(4)	Tool length compensation No.	Sets the tool length compensation No. used to shift the feature coordinate origin to the tool center position.
(5)	Input section	Displays the key entered to set "FeatShift amount" and "Tool L offset#".

Menus

Menus	Details
CutEdge-to org	This sets the current tool nose position to the feature coordinate origin.
Shift crd	This shifts the feature coordinate origin by the shift amount set in "FeatShift amount".
Set crd	This shifts the feature coordinate origin so that the value set in"FeatShift amount" is the current tool nose position.
Next axis	This switches the displayed axes in the inclined surface coordinate position and workpiece coordinate system offset to the next five axes. If the number of axes is 5 or less, this menu is grayed out and non-selectable.
Return	This displays the machining surface selection screen.

9.2.5.1 Shifting the Feature Coordinate Origin to the Tool Center Position

Operation method

(1) Machining target surface selection and indexing are performed on the machining surface selection screen.

For details, refer to "9.2.4.1 Selecting a Machining Target Surface", "9.2.4.2 Indexing the Machining Target Surface".

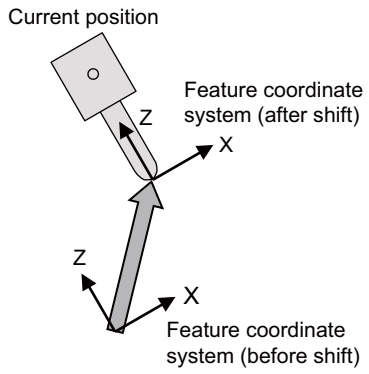
(2) Set the tool center at the position to be set as the feature coordinate origin, with manual operation.

(3) Press the menu [featcrdshiftin].

(4) Set the No. of tool length compensation amount in "Tool L offset#".

(5) Press [CutEdgeto org].

➡ The workpiece offset of the basic coordinate system on the selected machining surface is changed, and the feature coordinate origin is set again to the tool center position.



Z axis

Shift amount = Inclined surface coordinate (before shift) - Tool length compensation amount (absolute value)

Other than Z axis

Shift amount = Inclined surface coordinate (before shift)

Note

(1) When the parameter "#1287 ext23/bit1" (Inclined surface coordinate display) is "1" and during tool length compensation command, do not use "Tool L offset#" on the feature coordinate system shift screen, but use the tool length compensation No. during tool length compensation command to calculate the shift amount.

(2) Do not shift the feature coordinate origin during the G44 (offset in negative (-) direction) command. It may cause an abnormal operation.

(3) Set the tool center position at the position to be set as the feature coordinate origin.

(4) Do not shift the feature coordinate origin during the tool radius compensation command. It may cause an abnormal operation.

9.2.5.2 Shifting the Feature Coordinate Origin by the Specified Amount

Operation method

(1) Machining target surface selection and indexing are performed on the machining surface selection screen.

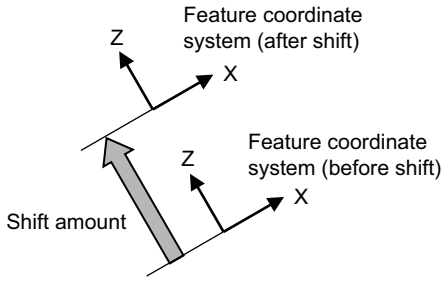
For details, refer to "9.2.4.1 Selecting a Machining Target Surface", "9.2.4.2 Indexing the Machining Target Surface".

(2) Press the menu [featcrdshiftin].

(3) Input the shift amount in "FeatShift amount" with a value on the feature coordinate system. Input "0" for an axis which is not to be shifted.

(4) Press the menu [Shift crd]. → The workpiece offset of the basic coordinate system on the selected machining surface is changed, and the feature coordinate origin is set again to the position shifted by the specified amount.

[Example of shift in the Z axis direction]



9.2.5.3 Shifting the Tool Center Position to the Position Specified on Inclined Surface Coordinate

The following shows an example to shift the feature coordinate origin so that the Z-axis inclined surface coordinate of the tool center is "10.".

Operation method

(1) Machining target surface selection and indexing are performed on the machining surface selection screen.

For details, refer to "9.2.4.1 Selecting a Machining Target Surface", "9.2.4.2 Indexing the Machining Target Surface".

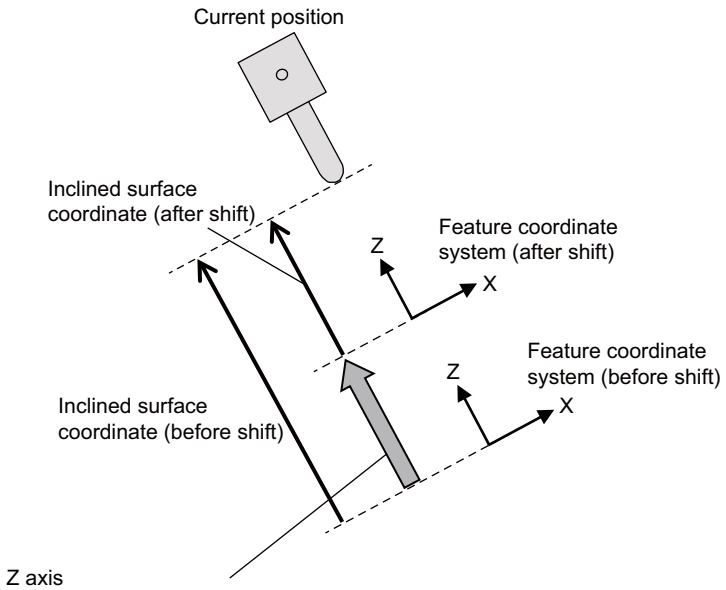
(2) Press the menu [featcrdshiftin].

(3) Set the desired inclined surface coordinate "10." in the Z axis of "FeatShift amount". Set the current feature coordinate in the X and Y axes which are not changed.

(4) Set the No. of tool length compensation amount in "Tool L offset#".

(5) Press the menu [Set crd].

➡ The workpiece offset of the basic coordinate system on the selected machining surface is changed, and the feature coordinate origin is set again so that the Z-axis inclined surface coordinate of the tool center is "10.".



Z axis
Shift amount = Inclined surface coordinate (before shift) - Inclined surface coordinate (after shift) - Tool length compensation amount (absolute value)
Other than Z axis
Shift amount = Inclined surface coordinate (before shift) - Inclined surface coordinate (after shift)

Note

(1) When the parameter "#1287 ext23/bit1" (Inclined surface coordinate display) is "1" and during tool length compensation command, do not use "Tool L offset#" on the feature coordinate system shift screen, but use the tool length compensation No. during tool length compensation command to calculate the shift amount.

(2) Do not shift the feature coordinate origin during the G44 (offset in negative (-) direction) command. It may cause an abnormal operation.

(3) Set the tool center position at the position to be set as the inclined surface coordinate.

(4) Do not shift the feature coordinate origin during the tool radius compensation command. It may cause an abnormal operation.

Precautions

(1) When the menu [CutEdgeto org], [Shift crd] or [Set crd] is pressed during single block stop, the offset of the basic coordinate system on the selected machining surface is changed; however, the feature coordinate system is not set again. In that case, make sure to command the inclined surface machining incremental multi-command (G68.4). The operation is performed on the feature coordinate system before workpiece offset change until G68.4 is commanded. The inclined surface coordinate counter may be a value different from the current position.

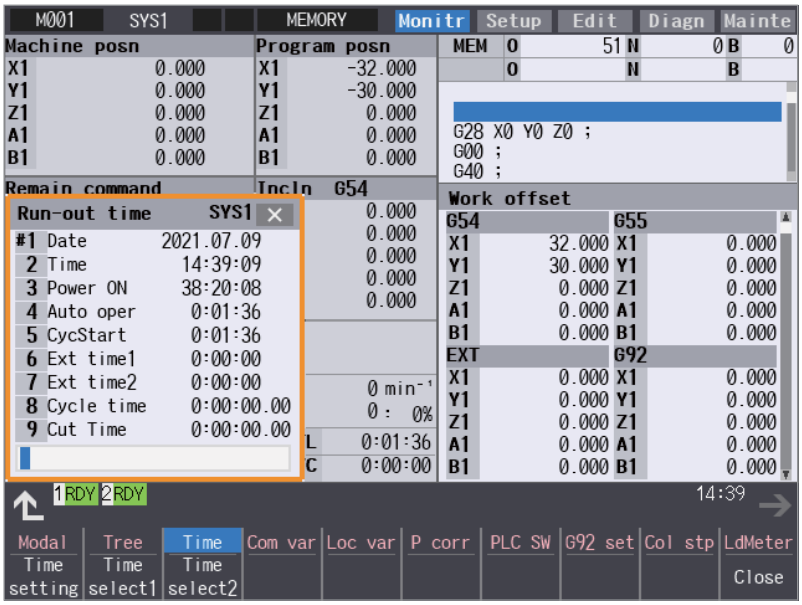
(2) When the "Manual absolute" (ABS) signal is OFF, the display of the inclined surface coordinate counter is not updated even with manual feed. When the menu [CutEdgeto org], [Shift crd] or [Set crd] is pressed even while the signal is OFF, the inclined surface G54 counter is updated.

9.3 Integrated Time

9.3.1 Integrated Time Display

Monitr – Time

The integrated time (date, time, power ON time, automatic operation time, automatic start time, external integrated time 1, external integrated time 2, cycle time and cutting time) controlled by the NC can be set and displayed.
Note that the cycle time cannot be set.
The integrated times displayed in cycle time display area on the Monitor screen can be set.



Display items

Display items	Details
#1 Date	The current date set in the NC is displayed. Year: 4 digits, Month: 2 digit, Date: 2 digit (YYYY.MM.DD)
2 Time	The current time set in the NC is displayed using the 24-hour system. (HH:MM:SS)
3 Power ON	This displays the total integrated time of the time from NC power ON to OFF. (HHHH:MM:SS)
4 Auto oper	This displays the total integrated time of the work time from automatic start button pressing in the memory (tape) mode or MDI to M02/M30 or reset button pressing. (HHHH:MM:SS)
5 CycStart	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 Ext time 1	This content differs depending on MTB specification. (HHHH:MM:SS)
7 Ext time 2	This content differs depending on MTB specification. (HHHH:MM:SS)
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.
9 Cut time	This displays the total time of the cutting commands (a part of the group 1 modal commands (*1)) given in the memory (tape) mode or MDI mode, from when the cycle start button is pressed until a feed hold stop, block stop, or a press of the reset button. This time will be cleared to "0" by pressing the cycle start button.
Input section	This displays details of the key input.

(*1) The following table shows the group 1 modal commands, the times of which are added to the cut time.

G01	Linear interpolation
G02/G03	Circular interpolation
G02.1/G03.1	Spiral/Conical interpolation
G02.3/G03.3	Exponential interpolation
G02.4/G03.4	3-dimensional circular interpolation
G06.2	NURBS interpolation
G33	Thread cutting
G34	Variable lead thread cutting
G35/G36	Circular thread cutting

The automatic return to the restart position when the program is restarted is not added to the cut time.
A travel command given during the direct command mode specification is added to the cut time.

Note

(1) When the indicated value reaches the maximum, the items #3 Power ON to #9 Cut time stops adding, and the maximum value remains displayed.

<Maximum time>
- #3 Power ON to #7 Ext time 2: 59999:59:59
- #8 Cycle time to #9 Cut time: 7999:59:59.99

Menus

Menus	Details
Time setting	This sets the integrated time.
Time select1	The time that is displayed in upper line on cycle time display area is selected.
Time select2	The time that is displayed in bottom line on cycle time display area is selected.
Close	This closes the pop-up window and quits this function.

9.3.2 Setting the Integrated Time

Operation method

(1) Press the menu [Time setting]. → The mode changes to the time setting mode. The cursor appears at the "#1 Date" position in the integrated time display.

(2) Input today's date.
(Example) 2021/4/1 [INPUT] → "2021.04.01" appears at "#1 Date", and the cursor moves to "#2 Time".

(3) Set the data for each item, and press the [INPUT] key.
If the item does not need to be set, press the [↓] key and move the cursor.
Note that "#8 Cycle time" and "#9 Cut time" cannot be set, so the cursor will not move. → When "#7 External integrated time 2" has been set, the cursor disappears, and the time setting menu highlight is turned OFF.

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 " / "	" : "

(2) If the menu [Time setting],  or  key is pressed again during the time setting mode, the mode will be canceled.

Setting range

Display items	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	

9.3.3 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.

(2) Press the time type for the mode to display.
(Example) [Auto operate] → 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.

(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) The displayed time is common for part systems.

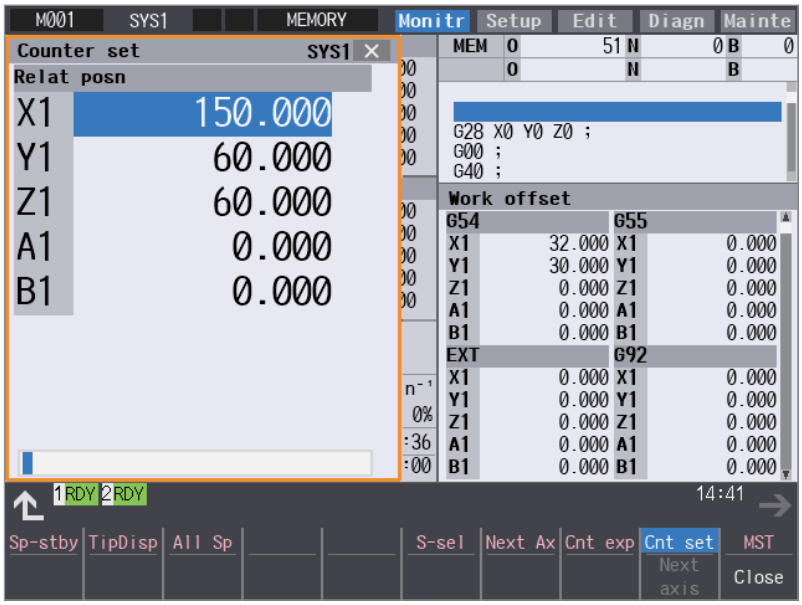
(3) The minimum unit for the cycle time and the cutting time is second.

9.4 Counter Setting

Monitr – Cnt set

Setup – Cnt 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.
It also can be set on the setup (Setup) screen.



Menus

Menus	Details
Close	This closes the pop-up window and quits this function.
Next axis	This switches the axis to be displayed on the counter. <Note> ♦This menu is grayed out and non-selectable in the following cases: - The number of enabled axes is 8 or less. - The number of enabled axes is 9 or more, however, the number of displayed axes is 8 or less according to the parameter "#1069 no_dsp Axis with no counter display" setting.

Operation method

(1)	Press the main menu [Counter set].	➡	The relative position counter opens as a pop-up window. The cursor appears at the 1st axis of the relative position counter.
	Instead of the operation above, the axis name address key can be pressed to set the counter. (Example) [Z]	➡	The relative position counter opens as a pop-up window. The cursor appears at the designated axis (Z axis).
(2)	Using the [↑], [↓], or the axis name address key, 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 control (cross axis control), the cursor moves to the axis corresponding to the address key to the axis name input first.
(3)	Input a numeric value. (Example) 100.000 [INPUT] If the [INPUT] key is pressed without inputting a value, zero (0) will be set.	➡	The set value "100.000" appears at the cursor position. The cursor moves to the next axis.
(4)	Repeat step (3) for each axis. For axes that are not to be operated, press the [↓] key and move the cursor.	➡	The pop-up window closes when the counter has been set for the last axis.

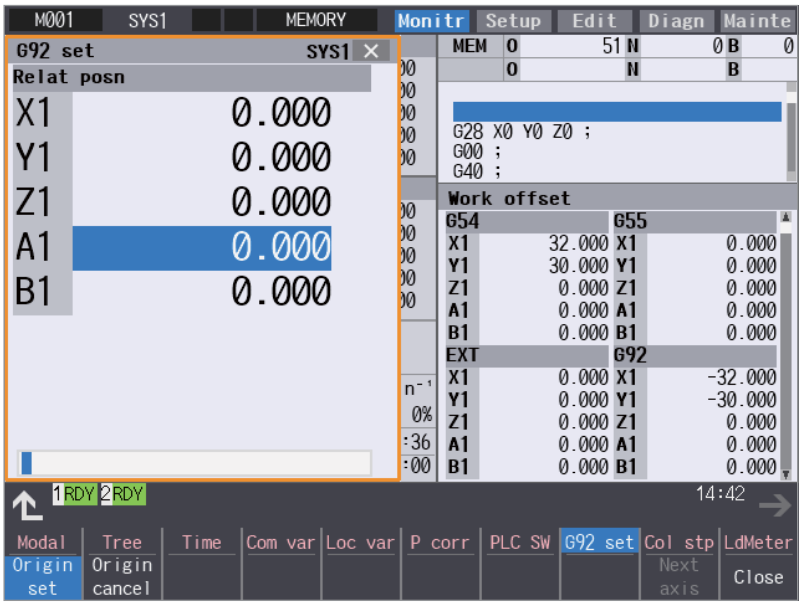
Note

(1) When an address such as "S", "M", "T" or "B" is used for the first character of axis name of the parameter "#1022 axname2", not the MSTB window but the Counter set window is displayed for address key input.

9.5 Origin Set, Origin Cancel

Monitr – G92 set

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



Menus

Menus	Details
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.
Origin cancel	The origin is canceled. The origin cancel mode is activated when this menu is pressed.
Next axis	This switches the axis to be displayed on the counter. <Note> ♦This menu is grayed out and non-selectable in the following cases: - The number of enabled axes is 8 or less. - The number of enabled axes is 9 or more, however, the number of displayed axes is 8 or less according to the parameter "#1069 no_dsp Axis with no counter display" setting.
Close	This closes the pop-up window and quits this function.

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 [G92 set]. → The relative position counter opens as a pop-up window. The menu [Origin set] is highlighted. The cursor appears at the 1st axis of the relative position counter.

(2) Using the [↑], [↓], or the axis name address key, 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 control (cross axis 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 [INPUT] key. If the [INPUT] key is pressed without inputting a value, zero (0) will be set. (Origin zero) → "0.000" appears at the cursor position. The cursor moves to the next axis.

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

Note

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

Operation method (Canceling the origin)

(1) Press the main menu [G92 set], and then press the menu [Origin cancel]. → The relative position counter opens as a pop-up window. The menu [Origin cancel] is highlighted. The cursor appears at the 1st axis of the relative position counter.

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

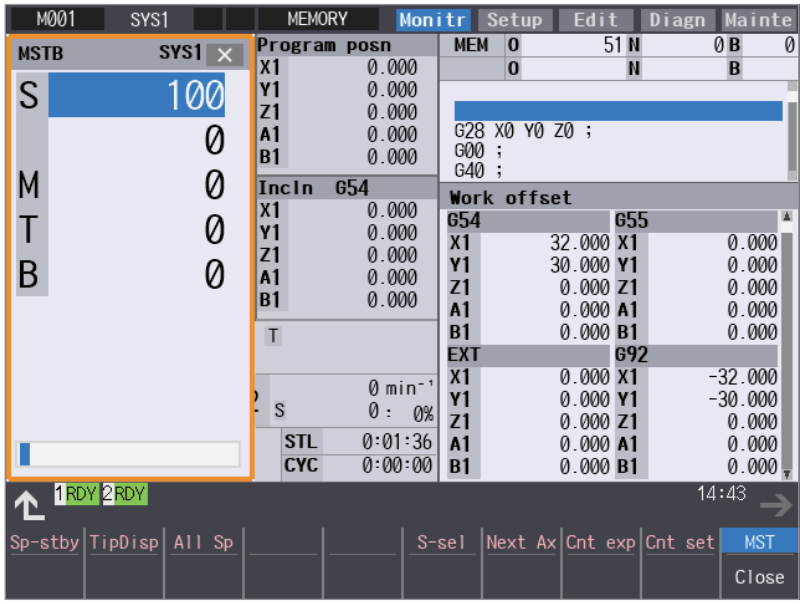
(3) Repeat step (2) for each axis. For axes that are not to be operated, press the [↓] key and move the cursor. → The pop-up window closes when the origin has been canceled for the last axis.

9.6 Manual Numerical Value Command

Monitr – MST

Setup – MST

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.
It also can be commanded on the setup (Setup) screen.



Display items

Display items	Details
S (spindle function) command value	The command value of S (spindle rotation command value) is displayed. The number of displays varies depending on the value of the parameter "#1039 spinno" (Number of spindles). However, if the value of the parameter "#1300 ext36/bit0" (Multiple-spindle control) is set to "1" (Multiple-spindle control II), the number of displays is set to "1" regardless of the value of the parameter "#1039 spinno" (Number of spindles). The spindle No. is shown or hidden depending on the parameter "#1725 cfg25/bit4" (Displaying spindle number on manual numerical value command screen). When the number of displays is "1", the spindle No. is not displayed.
M (miscellaneous function) command value	The command value of M (miscellaneous function command value) is displayed. The number of displays is set to "1" regardless of the value of the parameter "#12005 Mfig" (M count).
T (Tool function) command value	The command value of T (Tool function command value) is displayed. The number of displays is set to "1" regardless of the value of the parameter "#12009 Tfig" (T count). Only the tool number is displayed regardless of the value of the parameter "#11038 T disp type".
B (second (2nd) miscellaneous function) command value	The command value of B (second (2nd) miscellaneous function command value) is displayed. The number of displays is set to "1" regardless of the value of the parameter "#12011 Bfig" (B count). However, the value is displayed or hidden depending on the value of the parameter ("#1170 M2name" (2nd miscellaneous code)).

Note

(1) If the number of spindles is 5 or less, the scroll bar is not displayed.

(2) The part system name set with the parameter "#1169 system name" is displayed on the window title bar even when a value is set in the parameter "#12144 system name2". For the sub part system, the part system name set with the parameter "#12059 SBS_name" is displayed even when a value is set in the parameter "#12145 SBS_name2".

Menus

Menus	Details
Close	This closes the pop-up window and quits this function.

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

(1) Press the main menu [MST]. → The S, M, T, B display opens as a pop-up window.

Using the [↑] or [↓] key, move the cursor to the setting position. → The cursor moves.

(2) The manual numerical value command can also be issued by using the address keys instead of the operation above.
(Example) [T] → The S, M, T, B display opens as a pop-up window. The cursor appears at T.

(3) Input a numeric value.
31 [INPUT] → The input value is set.

Note

(1) 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 [INPUT] key.

- Press the menu [Close].
- Press the [◁] key.
- Change the screen.

Precautions

(1) The manual numerical value command cannot be executed while M, S, T or B is commanded.

(2) Even though the macro interruption commands (M96, M97) and subprogram call commands (M98, M99) are issued, they will not be executed.

(3) "0" is displayed on M field when M command has never been issued.

(4) When the block which includes multiple T commands or multiple the 2nd miscellaneous function commands is executed, the 1st command value to be valid is displayed on T or B field.

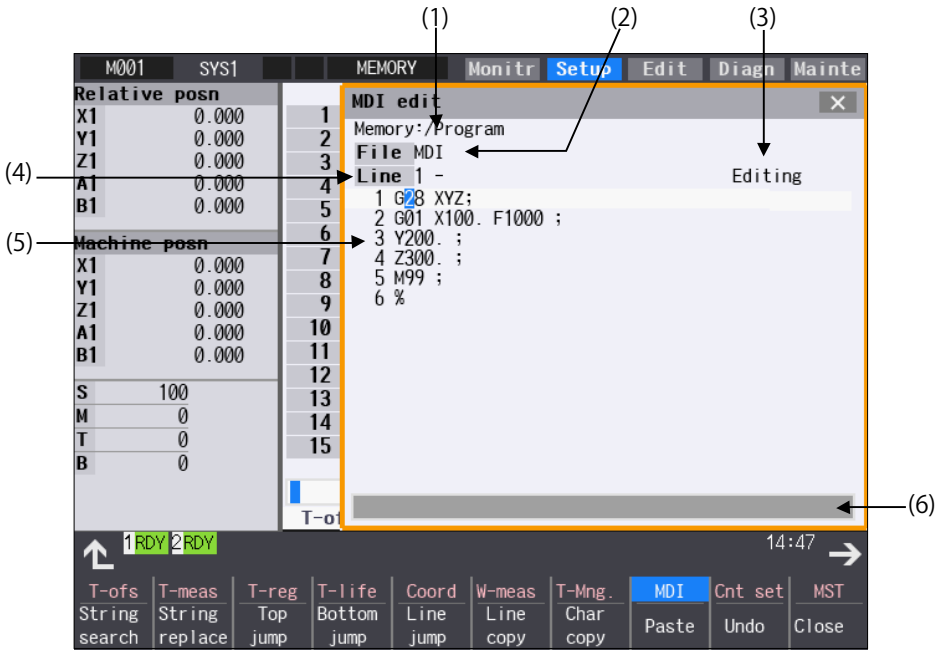
(5) When the block which includes a single M command is executed after the block which includes multiple M commands is executed, the values of the 2nd M command and later are not updated.

(6) When the parameter "#1264 set36/bit2" (Disabling manual numerical value command during automatic operation) is set to "1", the manual numerical value command cannot be executed during automatic operation.

9.7 MDI Program Editing

Setup – MDI

Press the main menu [MDI] 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].
For details on editing of MDI program, refer to "3.6 Editing Operations" and the following descriptions.



Display items

	Display items	Details
(1)	Path display	The path of the currently opened program file (memory:/program) is displayed.
(2)	File name display	This displays the file name of the program currently being edited (MDI).
(3)	Status	<Current edit display "Editing"> Displayed when the program is edited after it was displayed. <Insert mode display> "INS" appears when the mode is switched to overwrite mode by pressing the [INSERT] key.
(4)	Top line of the displayed program	The No. of the top line of the program is displayed.
(5)	Line number	<Line number> This displays the last 3 digits of the program line number. When over two or more lines are displayed on the screen because one line is long, the line No. does not attach to the beginning of the next line.
	Program display	<Program display> This displays the contents of the MDI program currently being edited.
(6)	Input section	This displays details of the key input.

Menus

Menus	Details
String search	When the character string is designated and the [INPUT] key is pressed, that character string is searched for.
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] key is pressed, the character string is replaced.
Top jump	The program data from the top line is displayed on the screen. The cursor moves to the left side of the top line.
Bottom jump	The program data from the last line is displayed on the screen. The cursor moves to the left side of the last line.
Line jump	When the line number is set and the [INPUT] key is pressed, the program data for the set number and later is displayed on the screen. The cursor moves to the No. of designated line.
Line copy	This copies the designated line (multiple lines can be copied.)
Char copy	This copies the character string designated by the cursor (multiple lines can be copied.)
Paste	This inserts the most recently copied contents to the line before the cursor or the position before the cursor.
Undo	This menu cannot be selected when parameter "#8910 Edit undo" is set to "0". <Note> ♦When the parameter "#8939 Undo confirm msg" is set to "1", the menu is highlighted while the confirmation message is displayed.
Close	This closes the pop-up window and quits this function.
Line clear	This erases the designated line (multiple lines can be erased).
Miss warning	This displays the input mistake warning for MDI program being edited.
Next miss	This moves the cursor to the next input mistake warning location.
Upward Tsearch	These menus search for T code upward or downward.
Down Tsearch	

Sub menus of [String search]

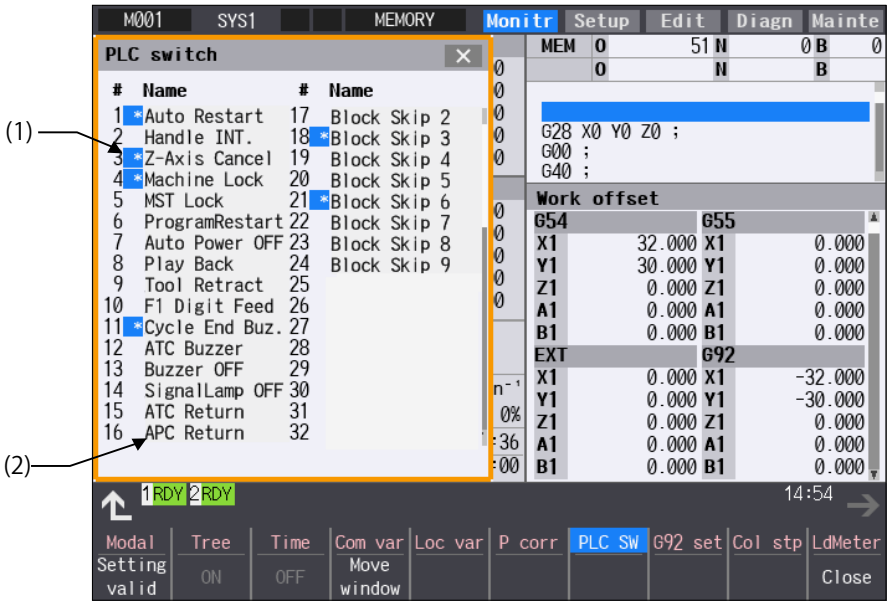
When the parameter "#19710 Edit-Upward search" is set to "1", the sub menus are displayed.

Menus	Details
Upward search	This switches the search direction upward.
Down search	This switches the search direction downward.
Quit search	This finishes the string search.

9.8 PLC Switch

Monitr – PLC SW

The various control signals for NC operation can be switched ON and OFF.
Refer to the instruction manual issued by the MTB for details.
The following screen is an example.



Display items

Display items		Description
(1)	Mark indicating switch ON	This is displayed for switches that are turned ON
(2)	Switch name	The display contents differ depending on MTB specification.

Menus

Menus	Description
Setting valid	Setting of the PLC switch is started. When the parameter "#8974 Simple PLC switch" is valid, this menu will be disabled.
ON	This turns ON the switch currently indicated by the cursor. (*1)
OFF	This turns OFF the switch currently indicated by the cursor. (*1)
Move window	This moves the display position of the PLC switch window. For the display unit other than 8.4- or 10.4-type, or when the parameter "#11019 2-system display" is set to nonzero, this menu cannot be selected.
Close	This closes the pop-up window and quits this function.

(*1) In the state that the PLC switch cannot be set, the menu is grayed out and made unselectable.
Also, when the cursor is at the PLC switch which cannot be set, the menu is grayed out and made unselectable.
Which PLC switch cannot be set is determined by each MTB.

9.8.1 Turning PLC Switches ON/OFF

Operation method (To turn switch "#6 ProgramRestart" ON)

(1) Press the menu [Setting valid]. → The menu Setting valid will be highlighted.
The message "Set up parameter?(Y/N)" is displayed.
Whenever the parameter "#8974 Simple PLC switch" is valid, the parameter setting will be enabled.

(2) Press the [Y] or [INPUT] key. → The PLC switch setting mode will be entered.
The cursor will appear at the PLC switch No. position.
The menus [ON] and [OFF] will appear normally.

(3) Press the cursor keys [↑] and [↓] to move the cursor to "#6". → The #6 number is highlighted.

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

(4) Press the menu [ON].

To turn OFF, press the menu key OFF. → The "#6 ProgramRestart" switch turns ON, and a [*] mark appears.

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

"*" mark appears.

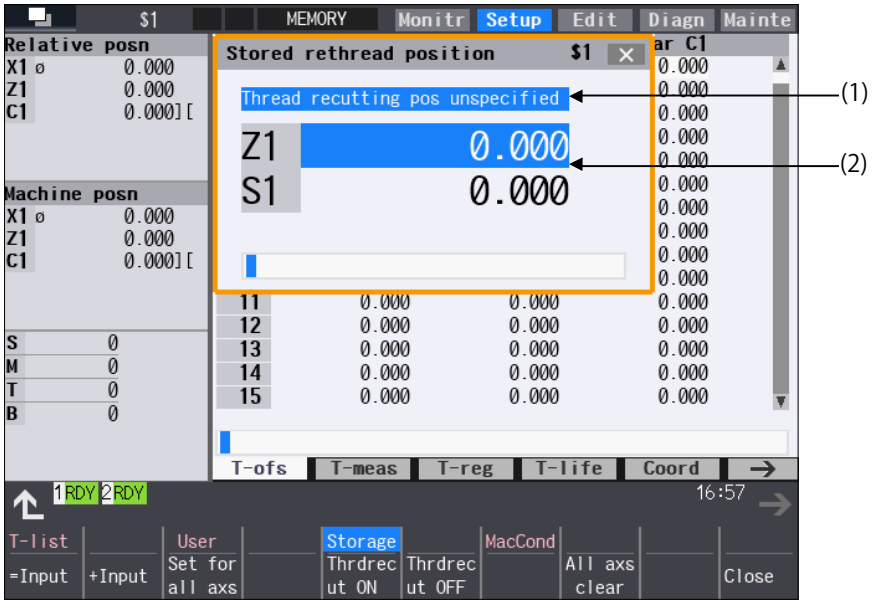
Note

(1) If the menu [Setting valid], ◁ or ↵ key is pressed again during the PLC switch setting mode, the mode will be canceled.
(2) The PLC switch setting mode is canceled when the PLC switch's pop-up window is closed.
(3) The cursor will not appear in modes other than the PLC switch setting mode.
(4) When the number of the switch is 64, the display range of the number can be switched between "# 1 to # 32" and "# 33 to # 64" by pressing the PAGE ▲ or PAGE ▼ key.

9.9 Thread Recutting Function

Setup – Storage

This function stores the thread groove position to enable thread re-machining.
Using this function allows the tool to pass through the machined thread groove by storing the lead axis machine position and the spindle angle to fit the groove position of the re-chucked workpiece.
This function is an optional specification.



Display items

Display items	Details
(1) Thread recutting status	Displays the status to suit the thread recutting machining status.
(2) Stored thread groove position	Displays the currently stored thread groove position. When the power is turned ON, "0" is displayed. Once the thread groove position is stored, it is held until the power is turned OFF or data is cleared.
Z1	Lead axis machine position The target axis follows the setting of the parameter "#8714 Thrdrecut lead ax".
S1 (*1)	Spindle angle The target spindle follows the setting of the parameter "#8715 Thread recut SP#".

Status	Details
Thread recutting pos unspecified	Displays the status in which the thread recutting position is not stored. When the menu [Thrdrecut ON] is pressed in this status, thread recutting is not enabled.
Thread recutting pos specified	Displays the status in which the thread recutting position is stored. When the menu [Thrdrecut ON] is pressed under this status, thread recutting is enabled. If thread cutting is commanded before the menu [Thrdrecut ON] is pressed, thread recutting is not carried out.
In thread recutting	Displays the status in which thread recutting is possible. When thread cutting is commanded under this status, thread recutting is carried out.

(*1) When a value is set in the parameter "#43107 S_disp_name" (Spindle display name), "S + spindle display name" is displayed according to its setting.

Menus

Menus	Details
=Input	This executes an absolute input.
+Input	This executes an incremental input.
Set for all axs	Press this menu to automatically input the lead machine position or spindle angle in the stored rethread position window and store data.
Thrdrec ut ON	This changes the status to "In thread recutting", enabling thread recutting using the stored data. If this menu is not pressed, thread recutting is not carried out. If this menu is pressed while thread recutting is disabled, the message "Thread recut position is unspecified" is displayed.
Thrdrec ut OFF	This changes the thread recutting status from "In thread recutting" to "Re-thread position specified", disabling thread recutting.
All axs clear	This erases all data of the stored positions.
Close	This closes the pop-up window.

9.9.1 Storing the Thread Groove Position

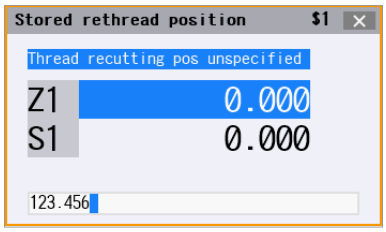
Operation method

Two methods are available to store the thread groove position: manual input and automatic input.

<Storing the thread groove position via manual input>

(1) Using the [↑] or [↓] key, switch the lead axis machine position (Z1) and the spindle angle (S1). → The cursor appears at the lead axis machine position (Z1) or the spindle angle (S1).

(2) Input the value in the item indicated by the cursor.
Z1: Lead axis machine position
S1: Spindle angle → The input value is displayed in the input area.

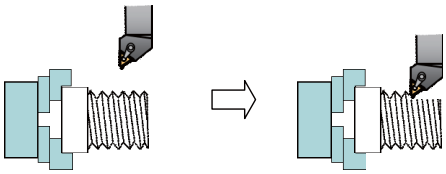


(3) Press the [INPUT] key or the menu [=Input].
<Note>
♦It is possible to perform incremental input by pressing the menu [+Input] key instead of the [INPUT] key or the menu [=Input].
(Example) If the original data is 200:
250 + [+Input] -> 200 + 250 = 450

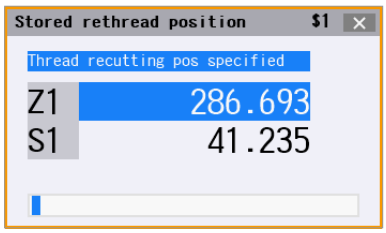
The input value is displayed in the input area.
The value is stored at the position indicated by the cursor, and the cursor then moves down. However, if the cursor is at the spindle angle (S1), press the [INPUT] key to move the cursor to the lead axis machine position (Z1).

<Storing the thread groove position via automatic input>

(1) Manually move the nose of the thread cutting tool to the thread recutting groove.



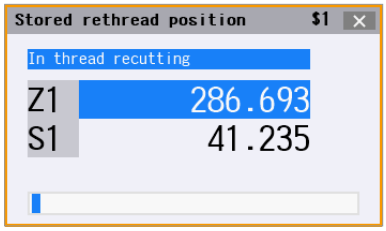
(2) Press the menu [Set for all axs]. → The thread cutting status is changed to "Thread recutting pos specified", and the current lead axis machine position (Z1) and spindle angle (S1) are stored collectively.



9.9.2 Switching the Stored Thread Groove ON/OFF

Operation method

(1) Press the menu [Thrdrecut ON] in the state "Thread recutting pos specified". → The thread cutting status is changed to "In thread recutting".
In this state, retract the tool and start the thread recutting program to enable the same thread machining as that carried out for the original thread.



(2) Press the menu [Thrdrecut OFF] in the state "In thread recutting". → The thread recutting status returns to the "Thread recutting pos specified" status, and thread recutting is disabled.

9.9.3 Erasing the Stored Thread Groove Position

Operation method

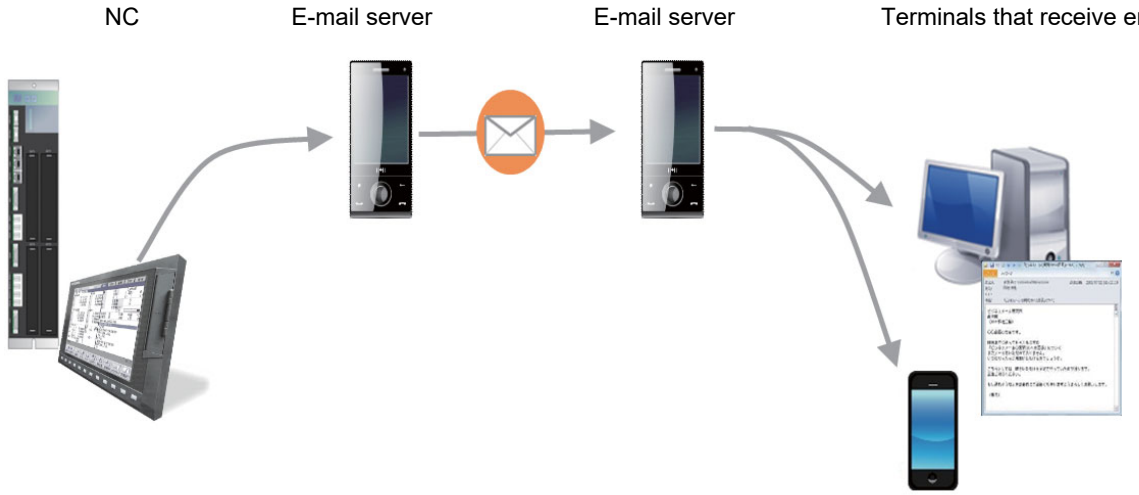
(1) Press the menu [All axs clear]. → The menu [All axs clear] is highlighted.
A message displays confirming whether it is okay to erase the data.

(2) Press the [Y] or [INPUT] key. → The lead axis machine position (Z1) or the spindle angle (S1) is cleared to "0".

9.10 Email Notification Function to Operator

Setup – 

This function enables NC to transmit emails to network-connected email servers (SMTP servers). The transmitted emails can be received by means of email software in PC or mobile phone via email servers. You are able to know machining conditions (such as machining completion, stop and failure) even if you are in a remote place.
"Email notification to operator" specification is required to use this function.



■Environment establishment

To use this function, NCs must be connected to the Internet. Connect LAN cable to NC units, and set the following parameters in accordance with rules of the connected network environment.
The communication data of this function is not encrypted. Locate the email server communicating with NC in a factory network.

Parameter No.	Item	Details
#1926	Global IP address	Set the NC IP address seen from an external source.
#1927	Global Subnet mask	Set the subnet mask for the IP address.
#1928	Global Gateway	Set the IP address for the gateway.

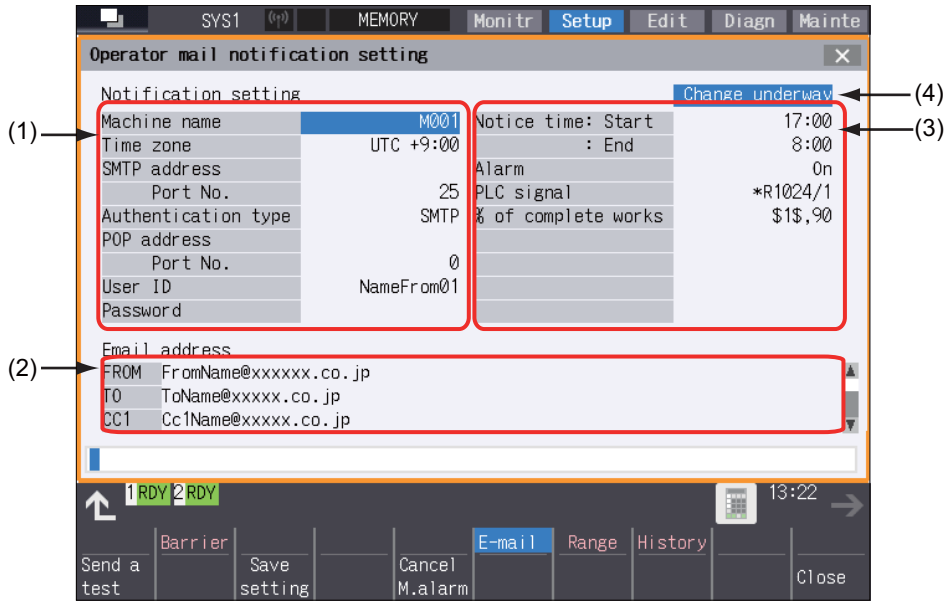
Note

(1) TCP/IP is only protocol usable for the LAN connection.

(2) The supported email server authentications are SMTP authentication and POP before SMTP authentication. Other authentications (such as Microsoft Exchange Server and SSL) are not supported.

9.10.1 Setting for Email Transmission Environment

Press the menu [E-mail] on the [Setup] screen to set the various items.



Display items

	Display items	Details
(1)	Email transmission environment setting	Set the items for email transmission environment.
(2)	Email address setting for transmission	Set email addresses of sender, recipient, etc.
(3)	Email transmission condition setting	Sets the conditions under which email transmission will be performed. It is optional whether you set "Notice time"; however, at least one item must be set (enabled) in "Alarm", "PLC signal" and "% of complete works" items. If none of these items are set, emails are not transmitted. An email is transmitted during automatic operation when an operating condition was satisfied with each of "Alarm", "PLC signal" or "% of complete works" setting items. Once an email was transmitted, there will be no email transmitted on the same setting item until automatic operation is completed. To send an email on the same transmission setting condition, automatic operation must be started after either machining program was completed or NC reset.
(4)	Change underway	This is displayed while one of the items (1) to (3) is being changed. Email notification stops while "Change underway" is displayed. To fix the change, press the menu [Save setting]. The confirmation message "Save the notification setting?(Y/N)" appears, then press the [Y] or [INPUT] key. "Change underway" disappears, and email notification will be enabled.

Note

(1) While in automatic operation, setting items cannot be changed on operator mail notification setting screen. If you attempt to change them, the error message "Executing automatic operation" is displayed. However, email notification setting information can be changed during single block stop.

(2) If you attempt to set items for email notification in the following cases, the error message "Data protect" appears.

- User's protection level is lower than that set by "Data protection by user's level" function.
- Data protection key 2 is OFF (0).

■Email transmission environment setting items and setting ranges

Setting item	Details	Setting range
Machine name	Set machine name displayed in transmitted emails. If not set, emails are sent with the machine name blank.	Alphanumeric (16 characters or less)
Time zone	Set time zone.	-12:00 to 14:00
SMTP address	Set SMTP server name for transmission. TCP/IPv6 is not available. You cannot set email servers only support SMTPS (e.g. Google email server).	Alphanumeric and symbols (64 characters or less)
SMTP Port No.	Set SMTP port number for transmission.	0 to 65535
Authentication type	Select authentication type from menu.	SMTP authentication/ POP before SMTP authentication
POP address	This item should be set only when you selected "POP before SMTP" authentication in "Authentication type". Set POP server name for receiving. This item is not necessary when you selected "SMTP" authentication (The setting will be ignored).	Alphanumeric and symbols (64 characters or less)
POP Port No.	This item should be set only when you selected "POP before SMTP" authentication in "Authentication type". Set POP port number for receiving. This item is not necessary when you selected "SMTP" authentication (The setting will be ignored).	0 to 65535
User ID	Set user ID in conformity with the email environment used.	Alphanumeric and symbols (80 characters or less)
Password	Set password in conformity with the email environment used. The input values are displayed in "*****".	Alphanumeric and symbols (24 characters or less)

[Setting change warning display]

When the setting of email address, user ID or password is changed, the operation alarm "M50 Email notif to opr setting chg" is displayed.
When this alarm is displayed unintentionally, change the setting of email address, user ID, or password on the email notification setting screen. The date and time when the email address for transmission was changed can be checked by confirming this alarm in the alarm history.

Note

(1) While another alarm is occurring, this alarm may not be displayed.
In that case, the occurrence log is kept in the alarm history.
(2) Check the alarm history regularly and confirm that an unintentional alarm does not occur.
(3) When the menu [Save setting] is pressed on the email notification setting screen, this alarm occurs regardless of the change contents.

For details of the alarm screen, refer to "11.1.1 Alarm History".

[About time zone setting]

Time zone is described in an email header at the time of transmission. If time zone is not included in the header, time of receipt may be displayed incorrectly depending of email software from which you receive the email. For this reason, set time zone in conformity with location in where machine is installed. If the time zone is not set, it is displayed in email headers as "0" (Greenwich Mean Time).

Example of time zone setting

Setting value	Example of standard time zone
+12	Kamchatka time, Russia - PETT
+11	Srednekolymsk Time, Russia- SRET
+10	Australian Eastern Standard Time - AEST Vladivostok Time, Russia - VLAT
+9:30	Australian Central Standard Time - ACST
+9	Japan Standard Time -JST Korea Standard Time - KST Yakutsk Time, Russia - YAKT
+8	Australian Western Standard Time - AWST Irkutsk Time, Russia - IRKT
+7	Krasnoyarsk Time, Russia - KRAT
+6	Omsk Time, Russia - OMST
+5:30	Indian Standard Time - IST
+5	Yekaterinburg Time, Russia - YEKT
+4	Samara Time, Russia - SAMT
+3	Moscow Time, Russia - MSK
+2	Eastern European Time - EET Kaliningrad Time, Russia - USZ1
+1	Central European Time - CET or MET
0	Greenwich Mean Time - GMT Western European Time - WET
-3:30	Newfoundland Standard Time - NST
-4	(The United States and Canada) Atlantic Standard Time - AST
-5	(The United States and Canada) Eastern Standard Time - EST
-6	(The United States and Canada) Central Standard Time - CST
-7	(The United States and Canada) Mountain Standard Time - MST
-8	(The United States and Canada) Pacific Standard Time - PST
-9	Alaska Standard Time - AKST
-10	Hawaii-Aleutian Standard Time - HAST(HST)

■Email address setting items

Setting item	Details	The number of registrable addresses
FROM	Set email transmission source address	1 (Required)
TO	Set recipient’s email address	1 (Required)
CC	Set email addresses to be carbon copied other than the recipient	8

Email transmission source address can also be set by inputting email transmission address file (MAILADDR.DAT) on I/O screen of the maintenance (Mainte) screen. For details, refer to "9.10.4 Inputting and Outputting Data".

Note

(1) Make sure to set the items "FROM" and "TO". If an email is transmitted without either one unset, alarm "Z39 E-mail send error" is issued.

■Email transmission setting items and setting ranges

Setting item	Details	Setting range
Notice time	Set hours for emails to be transmitted (start and end time). (Example 1) Start: 8:00 AM - End: 5:00 PM Emails are sent between 8:00 AM and 5:00 PM. (Example 2) Start: 5:00 PM - End: 8:00 AM Emails are sent between 5:00 PM and 8:00 AM of the next morning (Example 3) Start: 8:00 AM - End: 8:00 AM Emails are sent all day long.	00:00 - 23:59
Alarm	Select from menu whether to send emails when alarms are issued: Off (NOT send an email when an alarm is issued) On (Send an email when a predetermined alarm is issued). [Alarms for which emails are sent] •Operation errors (M) •Servo/Spindle alarms (S) •MCP alarms •System alarms (Z) •Emergency stop alarms (EMG) •Program errors (P)	Off/On
PLC signal	Set PLC signals for which emails are sent. (*1) (Example 1) X**0/1 In case of bit device, set device number to X**0 or X**8. Also, set bit number between 0 and 7. (When you designate X129, set as X128/1.) (Example 2) *R1024/1 "*" means B contact and "/1" means bit 1 of word device. (Example 3) 0 The settings are cleared.	Alphanumeric and symbols (64 characters or less)
% of complete works	An email is transmitted when the specified proportion of machining is completed. (Example 1) 1, 80 Set part system and the proportion of the number of workpieces machined ("#8002 WRK COUNT"/"#8003 WRK COUNT LIMIT"). If this is unset, it is regarded as $1. (Example 2) 0 The settings are cleared.	0 to 100

(*1) Usable devices and setting ranges are as follows:

Device	Setting range	Device type	Device	Setting range	Device type
X	X0 to X1FFF	Bit	SM	SM0 to SM16383	Bit
Y	Y0 to Y1FFF	Bit	V	V0 to V1023	Bit
M	M0 to M122879	Bit	SW	SW0 to SW7FF	Word
L	L0 to L2047	Bit	SD	SD0 to SD16383	Word
F	F0 to F4095	Bit	D	D0 to D8191	Word
SB	SB0 to SB7FF	Bit	R	R0 to R32767	Word
B	B0 to B1BFFF	Bit	W	W0 to W5FFF	Word

Menus

Menus	Details
Send a test	This executes email transmission test.
Save setting	This saves the email notification settings and enables the function again.
Cancel M.alarm	This cancels currently occurring alarm.
Close	This closes the pop-up window and quits this function.

9.10.2 Email Transmission Test

You can test email transmission to confirm that an email is send properly in accordance with the set conditions.

Operation method

(1) Press the menu [Send a test]. → The confirmation message "Send a test mail?(Y/N)" appears.

(2) Press the [Y] or [INPUT] key. → The operation message "Test mail is being sent." appears.
A test email is sent to all email addresses registered.
When the test email transmission is completed, the message "Test sending is complete." appears.

[Example of test email]

Title: Test message from M001 (Machine name)
Date & Time: 2016/5/27(Fri) 13:10
To: ToName@xxxxx.co.jp

(Message)
Test message has been sent.

Note

(1) If there is an error in notification settings or parameter "#8134 E-mail send disabled" is set to "1", and a test email is transmitted, message "Test sending error" is displayed and alarm "Z39 E-mail send error" is issued. To cancel the operation, press the reset key or select the menu "Cancel M.alarm" of operator mail notification setting screen.

(2) If wrong email address is set in "TO" or "CC" of "Email address" section and a test email is transmitted, message "Test sending is complete." is displayed. However, the test email is not send to the email addresses set in the "TO" or "CC". An email is transmitted from email server to the email address set in "FROM" to notify that the email transmission failed.

9.10.3 Description of Email Transmission Function

Emails are transmitted only during automatic operation (except test email transmission). When preparation is completed and email transmission conditions are satisfied during automatic operation, an email like a following example is sent to the registered addresses.
If you want to disable email transmission temporarily, set parameter "#8134 E-mail send disabled" to "1".

[Example of transmitted email]

Title: Message from M001 (Machine name)
Date & Time: 2016/5/27(Fri) 13:10
To: ToName@xxxxx.co.jp

(Message)
Below alarm is issued.
[Alarm message]
○○: Alarm number

Item	Details
Title	Name of source machine from which the email is transmitted. (Example) If "MACHINE001" is set in the machine name, the title will be "Message from MACHINE001".
Message	Alarm message Alarm number which is issued at the email transmission.

Note

(1) An email is transmitted if a condition for email transmission was already satisfied at startup of automatic operation.
(2) Even if an alarm that matches email transmission condition is issued during graphic check, an email is not transmitted.

9.10.4 Inputting and Outputting Data

You can input/output email transmission address settings as email transmission address file (MAILADDR.DAT).

To select the address settings as input/output target, press the menu [Dir] - [Email address] on I/O screen of the maintenance (Mainte) screen.

The directory and file name are automatically entered when the menu [Email address] is selected.

Device	Memory
Directory	Email transmission address
File name	MAILADDR.DAT (*1)

(*1) MAILADDR.DAT is a text file.

[Example of email transmission address file format]

```
[OPERATOR_MAILADDRESS]
FROM=FromName@xxxx.co.jp
TO=ToName@xxxx.co.jp
CC1=CC1Name@xxxx.co.jp
CC2=CC2Name@xxxx.co.jp
CC3=CC3Name@xxxx.co.jp
CC4=CC4Name@xxxx.co.jp
CC5=CC5Name@xxxx.co.jp
CC6=CC6Name@xxxx.co.jp
CC7=CC7Name@xxxx.co.jp
CC8=CC8Name@xxxx.co.jp
[OPERATOR_ENVIRONMENT]
USER_ID=NameFrom01
PASSWORD=12ABC#def@
```

Class	Key	Setting data	Data range
OPERATOR_MAILADDRESS	FROM	Transmission source email address	80 or less single-byte alphanumeric characters (lowercase/uppercase sensitive) and single-byte symbols
	TO	Destination email address	
	CC1 to 8	Email addresses carbon-copied	
OPERATOR_ENVIRONMENT	USER_ID	User ID	80 or less single-byte alphanumeric characters (lowercase/uppercase sensitive) and single-byte symbols
	PASSWORD	Password	24 or less single-byte alphanumeric characters (lowercase/uppercase sensitive) and single-byte symbols

Precautions

■Precautions for inputting data

(1) If space after "Key name (e.g. "FROM", "TO", etc.) =" is blank, the key data is cleared.

(2) Characters after ; (semicolon) in a line are treated as comments.

(3) In a case of an error (exceeding character limit, illegal key or format error), the subsequent portions are not input.

■Precautions for outputting data

(1) When the data file is unset, only key names ("FROM=", "TO=", etc.) are output.

■Precautions for data protection

(1) If you attempt to input/output the email transmission address file in the following cases, the error message "Data protect" appears.

- User's protection level is lower than that set by "Data protection by user's level" function.
- Data protection key 2 is OFF (0).

■Precautions for batch backups

(1) The email transmission address file cannot be backed up by batch backup function.

9.11 Checking the Settings Change History

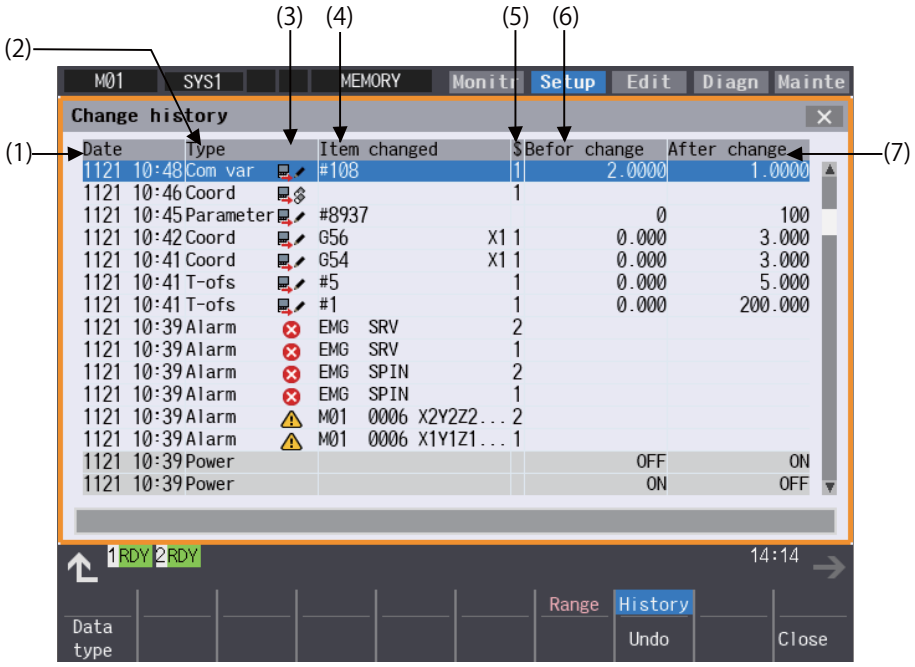
Setup – History

This screen shows the history of changes made to settings during setup operations for parameters, tool compensation, workpiece coordinate system offset, and other variables, listed in chronological order from newest to oldest. Settings can also be selected and restored from the change history list. Checking for missing settings during setup work or reviewing the change history can save time in checking setup.

Note

(1) The following items are not displayed in the settings change history:

- Name changes for common variables
- Changes of machine and safety parameters
- Setting changes from the ladder
- Changes to tool offset settings for the 2nd additional axis



Display items

<table>
<tr><th colspan="2">Display items</th><th>Details</th></tr>
<tr><td>(1)</td><td>Date</td><td>Displays the date of the change.
For 8.4-type/10.4-type: Displays in MMDD HH:MM format.
For 15-type/19-type: Displays in MMDD HH:MM:SS format.
The log is displayed in descending order of date and time.</td></tr>
<tr><td>(2)</td><td>Data type</td><td>Displays the changed data type.
If the data type is [Power supply], the background color is gray.</td></tr>
<tr><td>(3)</td><td>Changing method/ alarm type</td><td>Displays the changing method or alarm type with an icon.
The changing method is displayed when the data type is tool compensation, coordinate system, parameters, common variable, or PLC switch.
Different icons are displayed for external changes, such as from a peripheral tool, and for changes from within the NC unit.
<table>
<tr><th colspan="2">Icon</th><th rowspan="2">Changing method</th><th rowspan="2">Additional notes</th></tr>
<tr><th>NC</th><th>External</th></tr>
<tr><td></td><td></td><td>On-screen operation</td><td></td></tr>
<tr><td></td><td></td><td>Batch clear</td><td>Cleared multiple targets, such as clearing lines or clearing all.</td></tr>
<tr><td></td><td></td><td>Batch settings</td><td>Configured settings for multiple targets, such as initializing select machining conditions.</td></tr>
<tr><td></td><td></td><td>File input</td><td>Changed settings by file input/output.</td></tr>
<tr><td></td><td>-</td><td>Machining program</td><td>Made settings by G10, variable commands, or other machining program commands.</td></tr>
<tr><td>-</td><td>-</td><td>Reserved tool wear compensation reflection</td><td>Reflected the reserved tool wear compensation.</td></tr>
</table>
If the data type is [Alarm], there are two types of icons:  indicates an alarm, and  indicates a warning.</td></tr>
</table>

<table>
<tr><th colspan="2">Display items</th><th>Details</th></tr>
<tr><td>(4)</td><td>Target</td><td>This displays the # No., offset number, parameter number, and variable number of the PLC switch.
If multiple targets are changed simultaneously, the changed parameter range will be displayed.
<table>
<tr><th>Data type</th><th>Class changed</th><th>Display format</th></tr>
<tr><td>Alarm</td><td>Alarm</td><td>Alarm number, parameter 1, parameter 2
(Example) S01 0053 XYZCA</td></tr>
<tr><td>PLC switch</td><td>PLC switch input signal</td><td>PLC switch number
(Example) #11</td></tr>
<tr><td>Power supply</td><td>AC power input</td><td>-</td></tr>
<tr><td rowspan="5">Tool compensation</td><td>Compensation type 1</td><td>Offset number
(Example) #100</td></tr>
<tr><td>Compensation types 2/3</td><td>Class, offset number
(Example) Length dimension #100</td></tr>
<tr><td>File input</td><td>-</td></tr>
<tr><td>Line clear, compensation types 1/2</td><td>First compensation number-last compensation number
(Example) #100-#101</td></tr>
<tr><td>Line clear, compensation type 3</td><td>Class, first compensation number-last compensation number
(Example) Wear #100-#101</td></tr>
<tr><td rowspan="3">Coord</td><td>Workpiece offset</td><td>Class, axis number
(Example) G54 X1</td></tr>
<tr><td>File input</td><td>-</td></tr>
<tr><td>Clear all</td><td>-</td></tr>
<tr><td rowspan="4">Parameters</td><td>General parameters (user parameters only)</td><td>Parameter No.
(Example) #10000</td></tr>
<tr><td>Axis parameters (user parameters only)</td><td>Parameter No., axis name
(Example) #10000 X1</td></tr>
<tr><td>Multi-item parameters(*1) (user parameters only)</td><td>Parameter No., item name
(Example) #10000 Data type</td></tr>
<tr><td>File input</td><td>-</td></tr>
<tr><td rowspan="3">Common variable</td><td>Common variable</td><td>Variable number
(Example) #100000</td></tr>
<tr><td>Clear line</td><td>First variable number-last variable number
(Example) #100000-#100001</td></tr>
<tr><td>File input</td><td>-</td></tr>
</table></td></tr>
<tr><td>(5)</td><td>Part system number</td><td>When setting items for each part system, such as system-specific parameters and tool compensation, the part system number is displayed. If the data is common between part systems, this will be blank.</td></tr>
<tr><td>(6)</td><td>Before change</td><td>Displays the value before the change.
If the value exceeds the maximum number of displayable characters, part of the value will show as "...".</td></tr>
<tr><td>(7)</td><td>After change</td><td>Displays the value after the change.
If the value exceeds the maximum number of displayable characters, part of the value will show as "...".</td></tr>
</table>

(*1) Items such as data type, number of data entries, and display format for SRAM open parameters (#41001 to #41100), in which a single parameter can change multiple settings

Menus

Menus	Details
Data type	Sorts and displays the setting change history by data type. The setting here remains active even after rebooting the machine.
Undo	Can be used to restore the previous values for the data currently pointed to with the cursor.
Close	This closes the pop-up window and quits this function.

Sub-menu buttons under [Data type]

Displays the history of items for the highlighted menu button.

Menus	Details
Tool comp.	This changes whether to show or hide the tool offset change history.
Coord	This changes whether to show or hide the workpiece offset change history.
Param	This changes whether to show or hide the parameter change history.
Com var	This changes whether to show or hide the common variables change history.
PLC SW	This changes whether to show or hide the PLC switch history.
Power supply	This changes whether to show or hide the AC power input history.
Alarm	This changes whether to show or hide the alarms/warnings history.
Return	This exits the settings for showing or hiding history for each data type.

9.11.1 Changing the Data Type Displayed in the Settings Change History

Press [Data type] to either show or hide the history displayed on the screen.

(1) Press [Data type].

(2) Press [Tool comp.]. → The tool compensation history is hidden.
If the list is scrolled, the top of the list (most recent history) will display. The cursor line position with this function is retained.
If the cursor line is empty, the cursor moves to the last history.
If no history is displayed, the cursor is not displayed.

(3) Similarly, pressing the menu button for a data type will show or hide history from that data type. → The history of the highlighted data types is displayed, and the history of data types not highlighted is hidden.

9.11.2 Undoing Changes

Changes to the tool compensation, parameters, workpiece coordinate system offset, common variables, and PLC switch configuration data can be restored to their original values. Note that this feature is limited to data that has been modified by on-screen operation or by machining program commands.

(1) Press the [↑], [↓], PAGE, or PAGE key to move the cursor to the change to be undone.

(2) Press [Undo]. → The configuration data pointed to by the cursor will be restored to the previous value and the settings change history will be updated.
If the list is scrolled, the top of the list (most recent history) is displayed. The cursor line position with this function is retained.

Note

(1) If [Undo] is pressed with parameter "#8939 Undo confirm msg" set to "1", the menu button is highlighted and the operation message "OK?(Y/N)" appears. The program returns to the previous status by pressing [Y] or [INPUT] key. If other key is pressed, the operation is canceled. When the operation is canceled, the menu highlight returns to normal and the message is deleted.

(2) Parameters for password settings, such as the Ethernet parameter "#9713 Host1 Password", cannot be undone using [Undo].

(3) Common variables protected from setting operation by the parameters #12111 to #12114 cannot be undone even if the menu [Undo] is pressed. The message "Cannot return to origin" appears.

9.11.3 Displaying the Parameter Guidance

For parameter or alarm data types, pressing the [?] key will display guidance on that item.

(1) Press the [↑], [↓], PAGE, and PAGE keys to move the cursor to the desired item for guidance.

(2) Press the [?] key. → Guidance will be displayed when the cursor is over a parameter or alarm changed through on-screen operation or by a machining program command.
For all other cursor positions, alarm guidance for the current alarm is displayed.
If parameter guidance cannot be displayed, guidance for the first machining parameter is displayed.

10

Machining a Workpiece

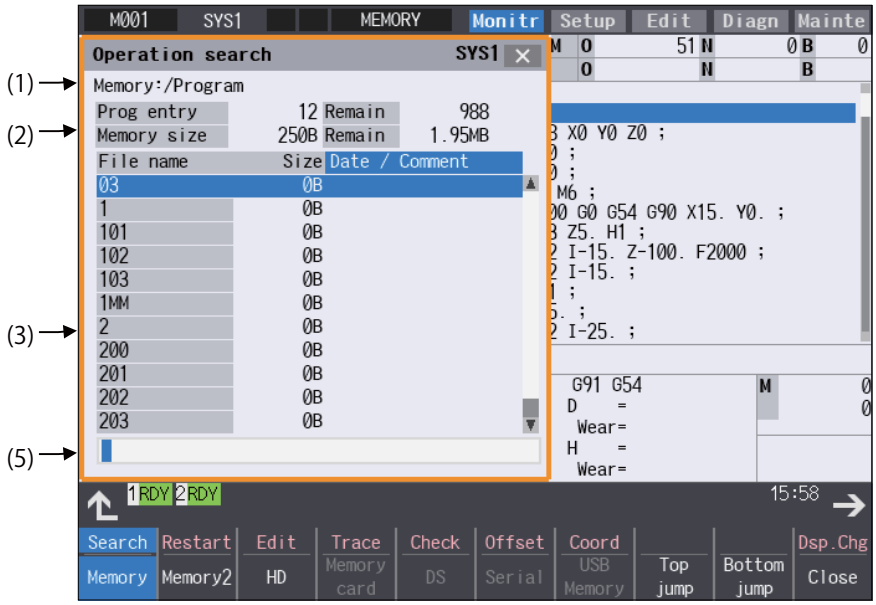
This chapter mainly describes the functions set on the monitor (Monitr) screen.

10.1 Operation Search

Monitr – Search

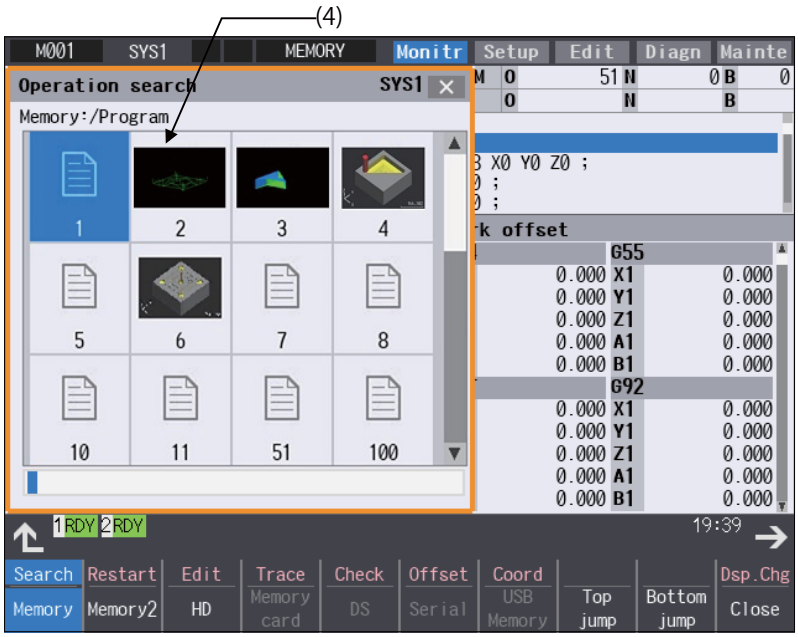
On this screen, the program can be called from the program storage site, such as a memory, by designating the program (program No.) to be automatically run and the program start position (sequence No., block No.).

■List display mode



■Thumbnail display mode

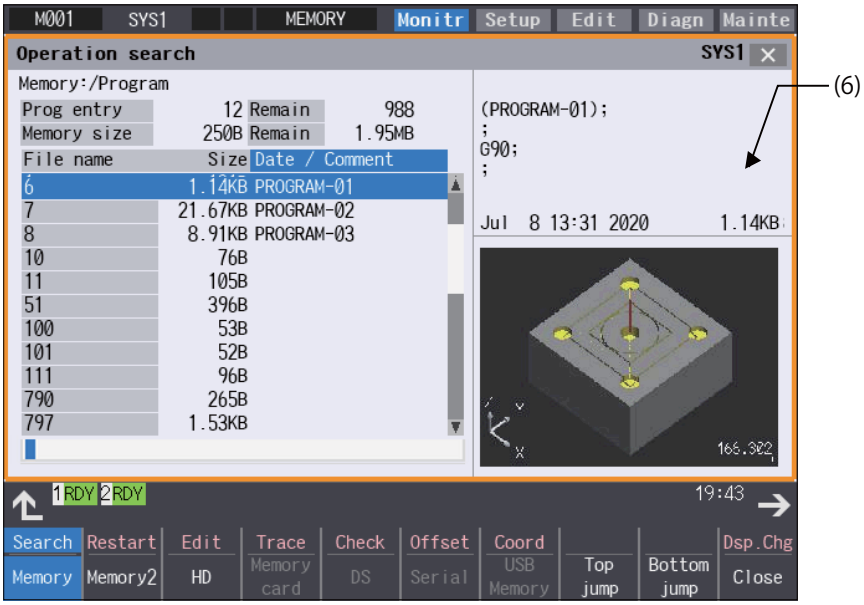
A list of thumbnails of image data related to the machining program can be displayed. Image data is pre-associated with the register image function. Press [Dsp. Chg] to switch between the list display and thumbnail display screens.



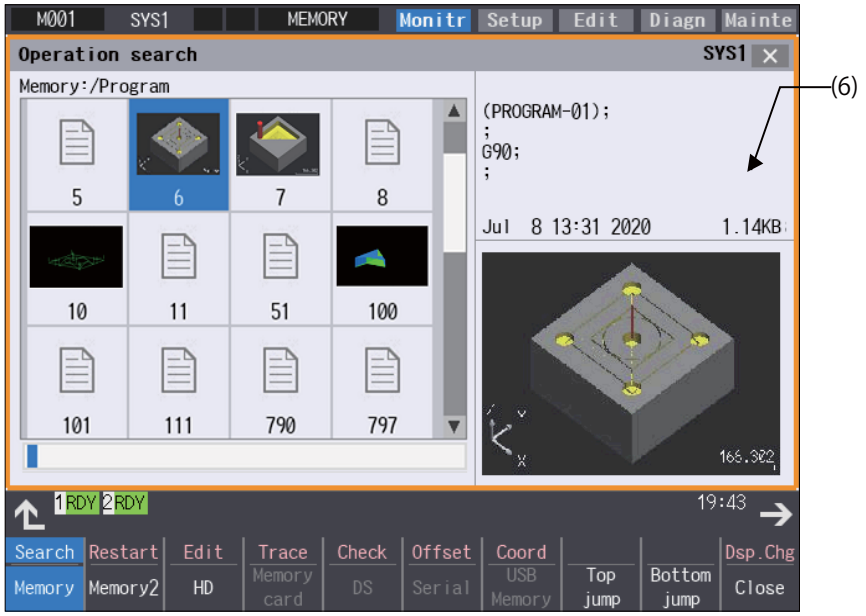
■Preview display mode

Press [Preview] to display a preview of the machining program selected in the list or by thumbnail, including a program description, associated image data, and the date and time of update.
Image data is pre-associated with the register image function.

<Preview from list display mode>



<Preview from thumbnail display mode>



Note

(1) If a serial device is selected, the description and other preview items for the machining program are displayed as empty.
(2) The preview displayed is the same for all supported screens.
(3) The display state is retained even after rebooting the machine.

Display items

	Display items	Details
(1)	Device name, directory display	This displays the device and directory where the searched machining program is located.
(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 [PAGE] and [PAGE] to scroll the displayed list. <Displaying "Date/comment" field> When the device is HD, memory card, DS or USB memory, the file update date/time is displayed. For NC memory, the display can be switched as follows depending on the parameter "#8981 NCmemory date disp" setting: (*1) When #8981 is set to "0", the machining program comment is displayed. When #8981 is set to "1", the file update date/time is displayed. Whether to show or hide the comment field can be selected by pressing [Comment nondisp]. 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)	Thumbnail display	Displays the file name of the machining program and a thumbnail of the image data associated with the machining program. If no image data is registered, the default image will be displayed. A maximum of 9 characters can be displayed for the file name. If the name exceeds 9 characters, an ellipsis "..." will appear after the 6th character.
(5)	Input section	This displays details of the key input.
(6)	Preview display	This displays details for the selected machining program. Description: A machining program is displayed from its top. A maximum of 4 lines can be displayed for 8.4/10.4-type display units and 16 lines can be displayed for 15/19-type display units. Display date: Displays the date and time which the file was updated. Display size: Displays the size. Display image: Displays the associated image data.

(*1) For the program list view of NC memory and NC memory 2, the program comment is displayed when the multi-part system program management is enabled (#1285 ext21/bit0=1), regardless of the parameter setting for "#8981 NC-memory date disp".

Menus

Menus	Details
Memory	This selects the device for searching for the program. When a device with directory is selected, the root is selected first.
Memory2	
HD	
Serial	
Memory card	
DS	
USB Memory	
Top jump	Display starts from the first page of the directory/list of files. The cursor moves to the top line.
Bottom jump	The last page of the directory/list of files is displayed. The cursor moves to the bottom line.
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.
Preview	This switches to show or hide the preview (detailed information).
Change Disp	This switches between list display mode and thumbnail display mode.
List update	This updates the list contents. (The latest contents of the currently selected device and directory are listed.)
Sort change	This changes the method of sorting the list.
Close	This closes the pop-up window and quits this function.

Note

(1) Depending on the model, devices available on the monitor screens are different.

Devices	Model	
	M800VW/M80VW Series	M800VS/M80V Series
NC memory	○	○
NC memory 2	○	○
HD	○	○ (*1)
Serial	○	○
Memory card (front-side SD card)	○	○
DS (built-in SD card)	○	○
USB memory	○	○

(*1) When the HD mode operation on an industrial computer (IPC) is enabled with M80V Series, the menu [HD] becomes selectable and you can select files in the hard disk on the industrial computer. The availability of this function depends on the MTB specifications (parameter "#1760 cfgPR10/bit2").
Set the IP address of the industrial computer to the parameter #11005 to use this function.
If the [HD] menu is not displayed, check the connection status of the industrial computer. Restart the industrial computer in the following cases.

- ♦When the CNC power is turned OFF and ON after connecting with the industrial computer.
- ♦When the screen saver or the auto sleep mode of the industrial computer has functioned.

(2) Some items may not be displayed depending on the device.
○: Displayed ×: Not displayed

Device/Display item	Memory	Memory2	HD	Serial	Memory card	DS	USB memory
Number of programs registered	○	○	○	×	○	○	○
Remain	○	○	×	×	×	×	×
Number of memory characters	○	○	○	×	○	○	○
Remain	○	○	○	×	○	○	○
List	○	○	○	×	○	○	○

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

10.1.1 Executing an Operation Search

Operation method

(1) Press the [$←→$] to select the part system to run. → The selected part system appears on the upper left of the screen.

(2) Press the main menu [Search]. → The sub-menu is displayed.
The list appears as a pop-up window.

(3) Select the device.
(Example) [Memory] → The selected device name and root directory (memory:/) are displayed in the device name and directory display fields.

(4) Press the [↑], [↓], [PAGE ▲], or [PAGE ▼] key to move the cursor to the directory containing the file to be set. → Moves into the directory.

(5) Press the [INPUT] key.
If the list contents differ from the actual device or directory, press the menu [List update].

(6) Press the [↑], [↓], [PAGE ▲], or [PAGE ▼] key to move the cursor to the target machining program.

(7) Press the [INPUT] key. → The search starts.
When the search is completed, the searched device and program position appear in the display area for the machining program currently being executed.

When designating the sequence number and block number and searching for the program, input "program No./sequence No./block No." 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) The maximum number of digits is eight for the sequence No. and nine for the block No.
(2) Program after operation search
(a) Operation search is canceled 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 mode during MDI operation even if the block is stopped. Wait for MDI operation to end or reset the NC before searching.
(3) A program with 33 or more file name characters cannot be searched.

(4) Memory card, DS and USB memory are in the device access state during the operation. Do not remove the memory card, DS and USB memory during its operation. Failure to observe this could result in faults.

(5) Searched program No. is memorized in NC even after the power is turned OFF. When NC starts up after the power is turned ON again, the program which has the previously searched program No. will be operated.

(6) In case the USB memory is changed to another one, if the program has the same No. as previously searched program exists in the currently inserted USB memory, the program will be operated. When the program of the same No. does not exist in the USB memory, the program error (P232) will occur.

(7) When you perform an operation search by designating a block No., it can be set by the parameter "#1760 cfgPR10/bit5" whether the block which consists of comment only is counted as one block. When the parameter is set to "0", the comment block is counted as one block.
Set the parameter to "1" to count the comment block as same as the automatic operation.

(Example) When an operation search is performed by the sequence No. 1 and the block No. 2.

Sequence Nos. and the block Nos. vary as indicated in the following table depending on the setting of the parameter. Thus, the different block is searched.

When "#1760 cfgPR10/bit5" is set to "0", the block (a) is searched.

When "#1760 cfgPR10/bit5" is set to "1", the block (b) is searched.

Program	#1760 cfgPR10/bit5 = 0		#1760 cfgPR10/bit5 = 1	
	Sequence No.	Block No.	Sequence No.	Block No.
N01;	N1	B0	N1	B0
(COMMENT1);	N1	B1	-	-
G28 X0Y0Z0; .....(a)	N1	B2	N1	B1
G0 X10.; .....(b)	N1	B3	N1	B2
M30;	N1	B4	N1	B3

(8) If the setting of the parameter "#11005 PC IP address" does not match the IP address of non-Windows-based display unit (M800VW/M80VW Series) or the IP address of an industrial computer (IPC) (M80V Series), "Search error" appears when you access a program stored in the display unit or the drive on IPC (e.g. USB memory).

(9) When a part system under automatic operation, program restarting, or program checking is searched, the message "Executing automatic operation", "Program restarting", or "Program checking" is displayed.

■Operation search when the multi-part system program management is enabled

When the following conditions are met, the program search is executed across all part systems in batch.

- The parameter "#1285 ext21/bit0" is set to "1" (Multi-part system program management is enabled.)
- The parameter "#1285 ext21/bit1" is set to "1" (Search through a program for all part systems at once is enabled.)
- Machining program stored in NC memory or NC memory 2

Operation search is performed in each part system as shown in the table below depending on the parameter "#8144 OP srch NB all" and whether or not the program No. (or file name) is specified.

#8144 Setting value	Specification of program No.	Displayed part system	Non-displayed part system
0	Specified	The block of the specified sequence No./ block No. is searched in the program with the specified No.	The head of the program with the specified No is searched.
	Not specified	The block of the specified sequence No./ block No. is searched in the program with the currently selected No.	The currently selected program No., sequence No. and block No. are held. (Search is not performed.)
1	Specified	The block of the specified sequence No./ block No. is searched in the program with the specified No.	The specified sequence No./block No. is searched in the program with the No. specified in the displayed part system.
	Not specified	The block of the specified sequence No./ block No. is searched in the program with the currently selected No.	The block of the specified sequence No./ block No. is searched in the program with the currently selected No. (*1)

(*1) For part system where a machining program with different program No. and device from those in the displayed part system is searched, the message "Search error" appears.

Note

(1) Operation search is not available in the sub part system of sub part system control I/II on M80V Series.

(2) Operation search is not available in the sub part system of sub part system control II on M800V Series.

(3) For a program where the timing synchronization command is used, do not execute operation search in the block with the timing synchronization command and the subsequent blocks. The block execution timing is off between the part systems, and the machine is in danger of collision.

10.1.2 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.

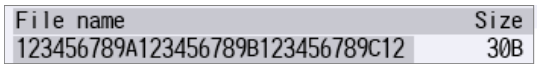
(2) Press the highlighted [Comment nondisp] again. → The comment field appears, and the file name is displayed up to 13 alphanumeric characters and symbols.

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

- When the comment field is displayed: The file name display is omitted and "*" is attached.



- When the comment field is hidden: The file name is displayed up to 32 alphanumeric characters and symbols.



Note

(1) The set status is held even if the NC power is turned OFF.
(2) The set status is common for list view of operation screen, restart search screen, etc.
(3) The cursor moves to the head when display/non-display in the comment field is switched.

10.1.3 Changing the Sorting Method

- Sorting method changes to 1 -> 2 -> 3 -> 4 -> 5 -> 1……every time the menu [Sort change] is pressed.
- When the sorting method 1, 2, or 3 is selected, "File name" column is highlighted. The sorting method 4 or 5 is selected, "Date/Comment" column is highlighted. ("1. File name No. ascending order" is selected by default setting.)
- The method is common for all the devices.
- You can sort up to the number shown in the table "Sortable number of files" below (the total number of files and directories, including current directory "." and one directory above "..") within a directory.
- "List updating" is displayed in flickering during sorting.
- If it exceeds the sortable number of files, the menu [Sort change] is grayed out and changing the sorting method is impossible. In this case, a program in NC memory or NC memory 2 is displayed with "6. Program No. ascending order". Program in other devices is displayed with "2. File name character code ascending order". In this case, "File name" column is highlighted regardless of the sorting method.

Sorting method

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
6	Program No. ascending order	1. Program No. (Numeric character) ascending order 2. Program No. (Character code) ascending order

Note

(1) The selected sorting method is held even after the power is turned OFF.

(2) The selected sorting method is common for list view of the monitor screen, restart search screen, etc.

(3) 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 to A because "1" equals to "0x31" and "A" equals to "0x41".

(4) When NC memory or NC memory 2 is selected as a device, display for "current directory" and "one directory above" is not available.

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

(6) Comment names (dates) are compared using character code only. Thus, the order will not be one by the actual date.

(7) When the number of the programs in the NC memory or NC memory 2 exceeds the sortable number of files, the programs are displayed with "6. Program No. ascending order".

Sortable number of files

Device	M800VW/M80VW Series	M800VS Series	M80V Series
Memory	128	128	128
Memory2	128	128	128
HD	1000	-	128 (*2)
Memory card	1000 (*1)	250 (*1)	128
DS	1000 (*1)	250 (*1)	128
USB memory	1000 (*1)	250 (*1)	128

(*1) The available sort volume can be designated using the parameter "#8937 File sort volume".

(*2) When the HD mode operation on an industrial computer (IPC) is enabled with M80V Series, you can sort the files in HD device. The availability of this function depends on the MTB specifications (parameter "#1760 cfgPR10/bit2"). Set the IP address of the industrial computer to the parameter #11005 to use this function.

10.2 Checking the Movement under Machining (Monitor Screen)

Monitr

Various information related to operation, such as the axis counter, speed display and MSTB command are displayed on the monitor screen. In addition, you can perform the following operations on the monitor screen.

- Search
- Research
- Editing the searched machining program
- Check (Display of NC program's tool movement path)
- Trace (Display of machine movement path)
- Correction of operating program's buffer
- Counter set
- Manual numerical value command

10.2.1 Screen Layout

The screen layout will depend on the size of the display unit, the number of displayed part systems, and the number of axes. For this section, the following screens are provided as examples to describe the display items.

No. of displayed part systems	Display unit size	Display status
1-part system	8.4-type/10.4-type	Normal view (up to 5 axes)
	8.4-type/10.4-type	Normal view (if selectable display is enabled)
	8.4-type/10.4-type	Simple view (up to 5 axes and input units are 0.01 μm or greater)
	15-type/19-type	Single counter
2-part system	8.4-type/10.4-type	Dual counter (up to 4 axes)
	15-type/19-type	Enlarged counter
	15-type/19-type	Dual counter (up to 5 axes)
3-part system	15-type/19-type	Dual counter (up to 5 axes)
4-part system	15-type/19-type	Quad counter

To display multiple part systems simultaneously, set the Parameter "#11019 2-system display" (2-system simultaneous display).

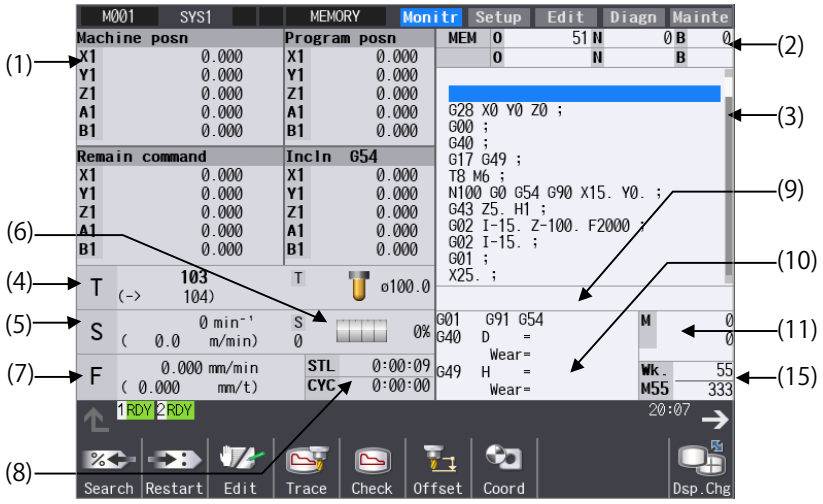
Setting value	Display
0	1-part system display
1, 2	2-part system simultaneous display
3	3-part system simultaneous display (15-type/19-type display units only)
4	4-part system simultaneous display (15-type/19-type display units only)

■1-part system display

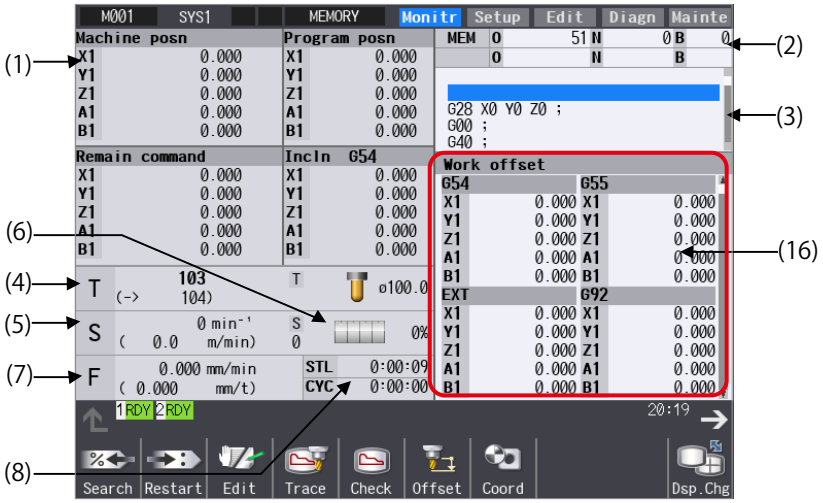
With 8.4-type/10.4-type display units, press [Simple view] or [Normal view] under [Dsp.Chg] to switch the display pattern.

With 15-type/19-type display units, press [Single counter], [Dual counter], or [Quad counter] under [Dsp.Chg] to switch the display format.

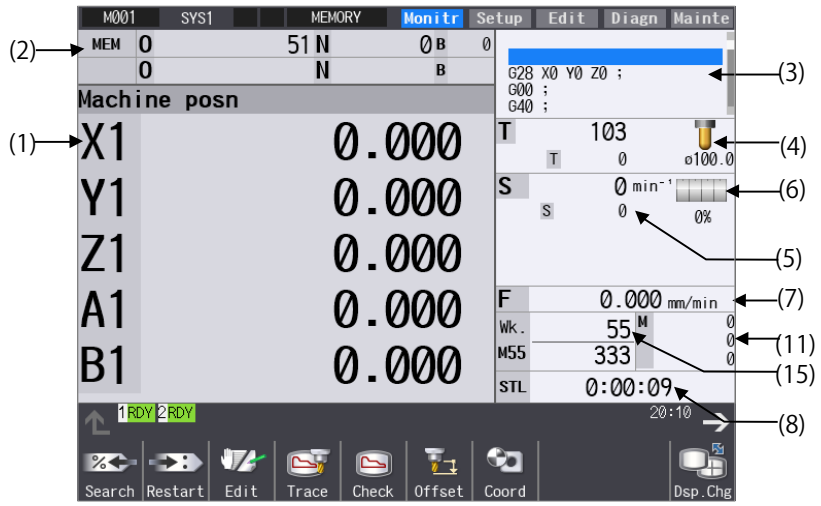
[8.4-type/10.4-type display units: Normal view (up to 5 axes)]



[8.4-type/10.4-type display units: Normal view (If selectable display is enabled)]

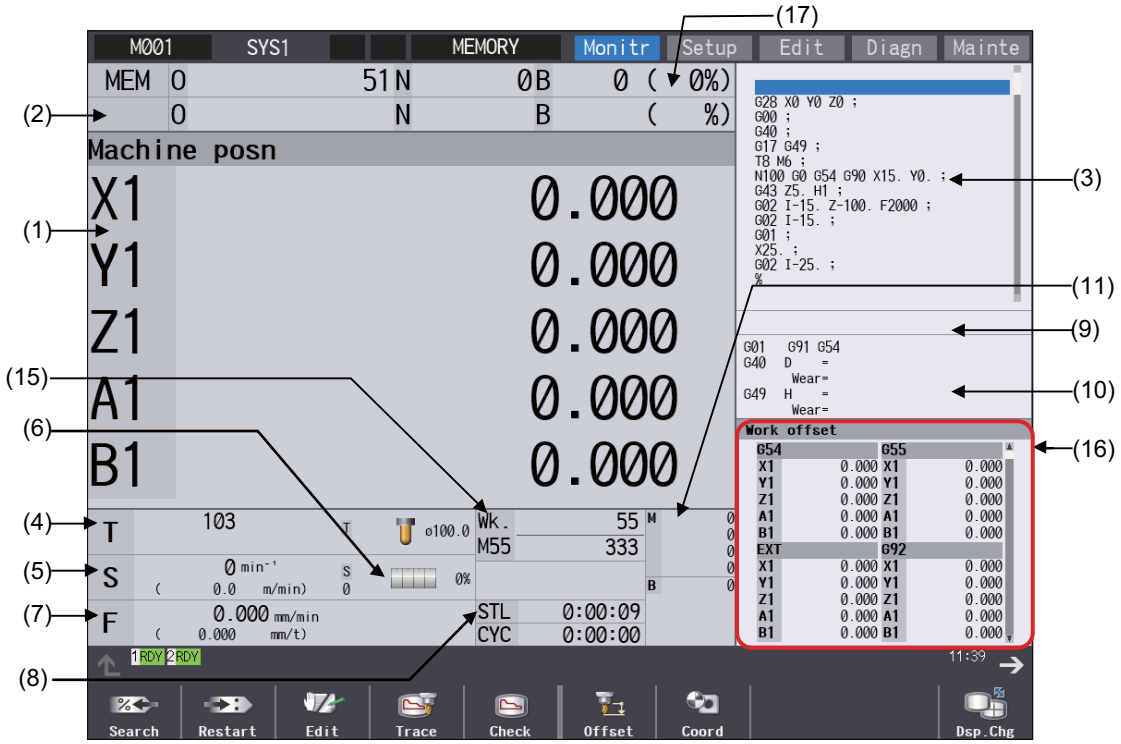


[8.4-type/10.4-type display units: Simple view (up to 5 axes and input units are 0.01 μm or greater)]



When the number of axes is 6 or more or the input units are 0.01 μm or less in any one of the part systems, the number of displayed counter digits will change for all part systems. In this case the maximum number of axes that can be displayed are 8 axes.

[15-type/19-type display units: Single counter]



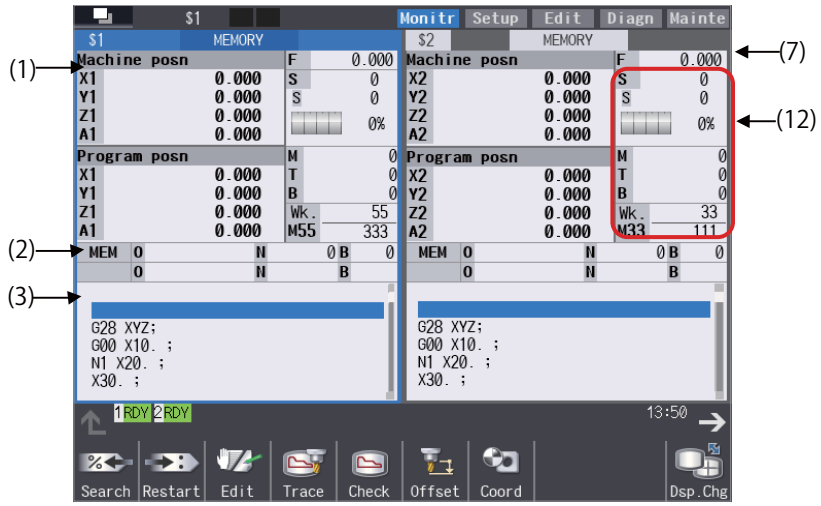
Note

(1) The maximum number of axes that can be displayed is 5 axes. Press the menu [Next axis] to switch display to display the subsequent axes.

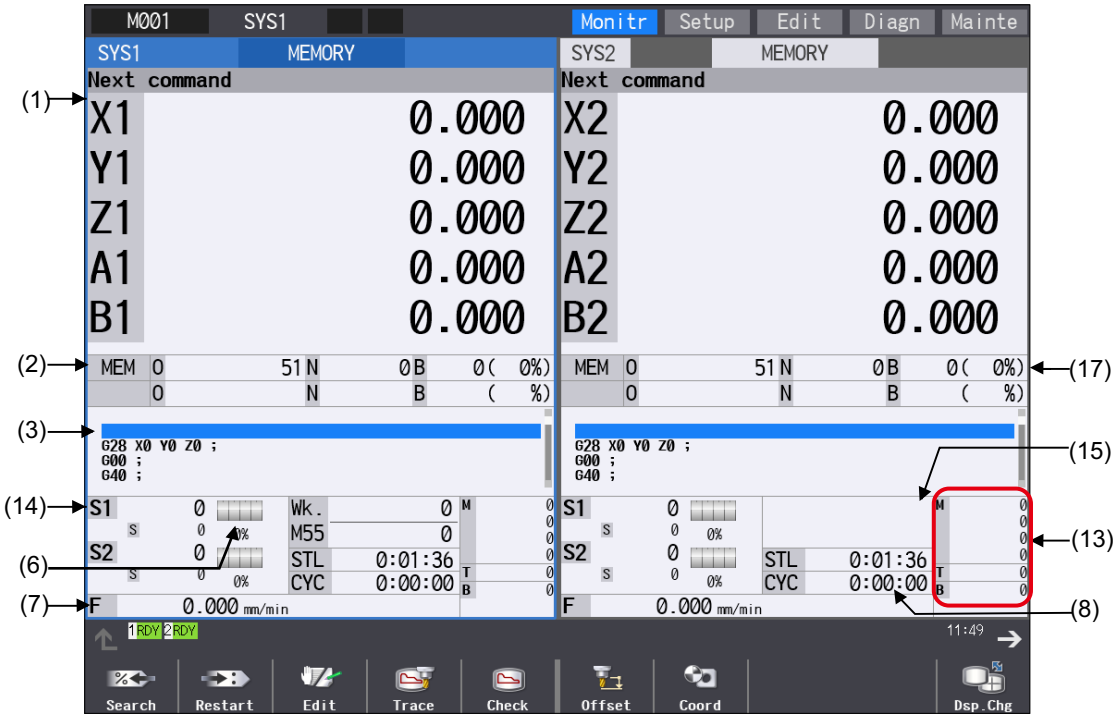
■2-part system simultaneous display

Press [Counterexpand], [Dual counter], or [Quad counter] under [Dsp.Chg] to switch the display format.

[8.4-type/10.4-type display unit: Dual counter (up to 4 axes)]



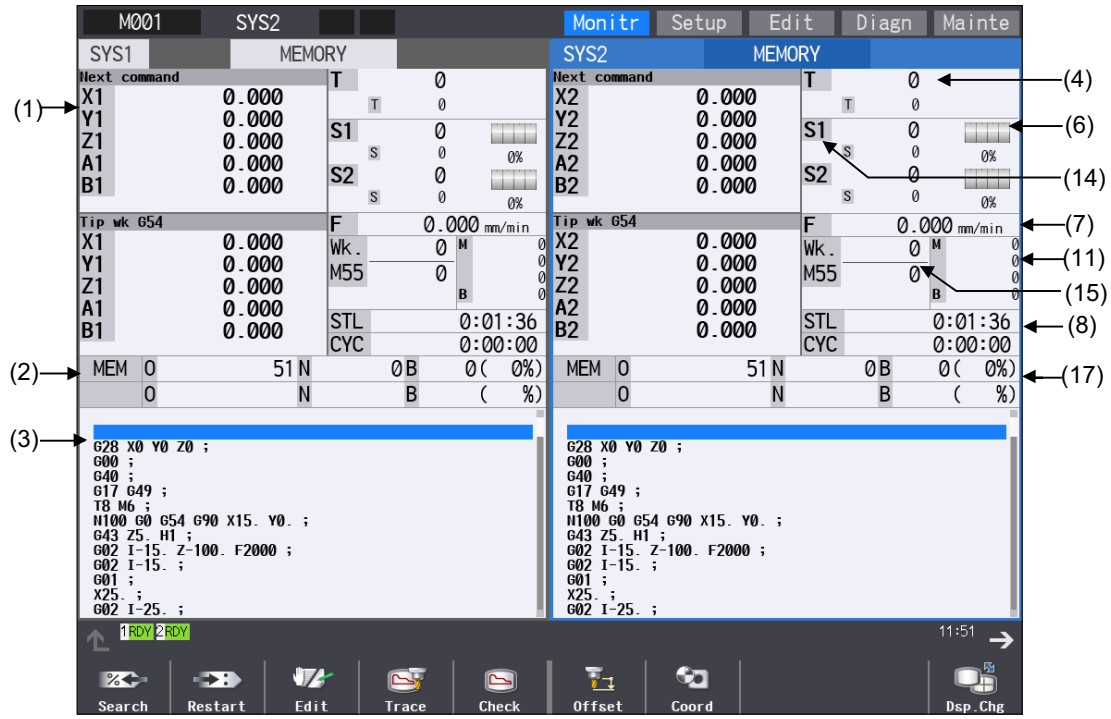
[15-type/19-type display unit: Counterexpand]



Note

(1) The maximum number of axes that can be displayed is 5 axes. Press the menu [Next axis] to switch display to display the subsequent axes.

[15-type/19-type display unit: Dual counter (up to 5 axes)]



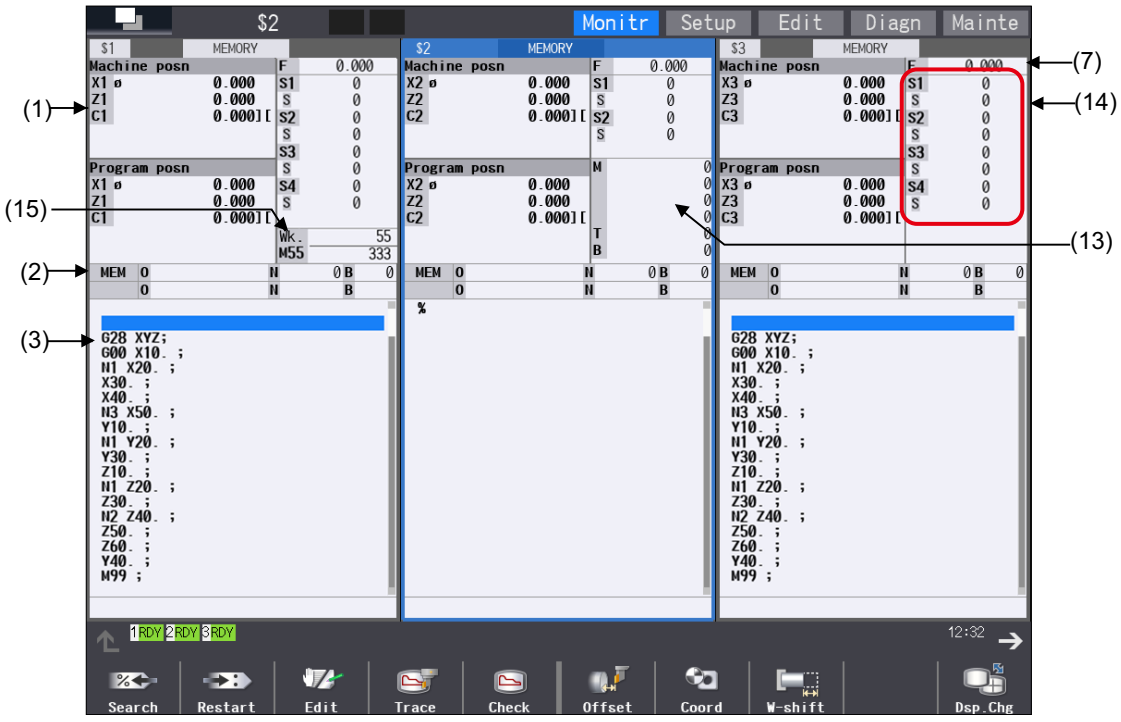
Note

(1) The maximum number of axes that can be displayed is 5 axes. Press the menu [Next axis] to switch display to display the subsequent axes.

■3-part system simultaneous display

Press [Counterexpand], [Dual counter], or [Quad counter] under [Dsp.Chg] to switch the display format.

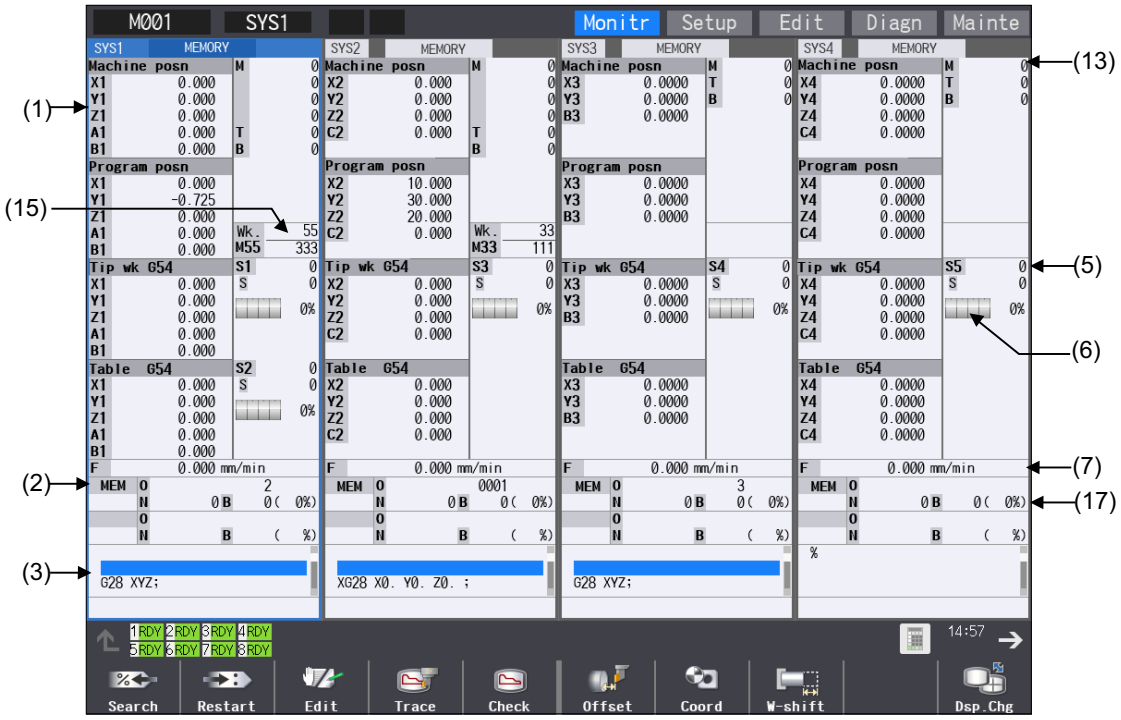
[Dual counter (up to 5 axes)]



■4-part system simultaneous display

Press [Counterexpand], [Dual counter], or [Quad counter] under [Dsp.Chg] to switch the display format.

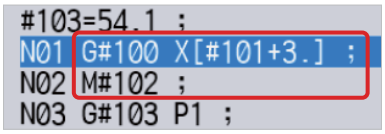
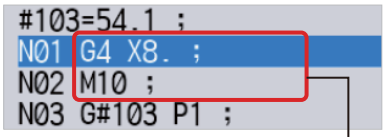
[Quad counter]

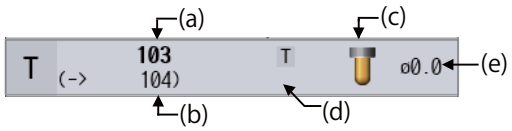
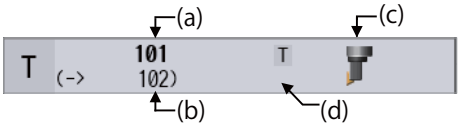
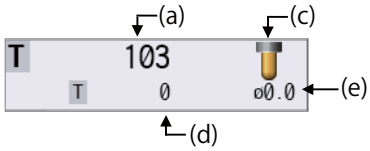
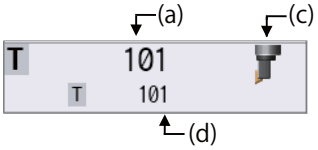


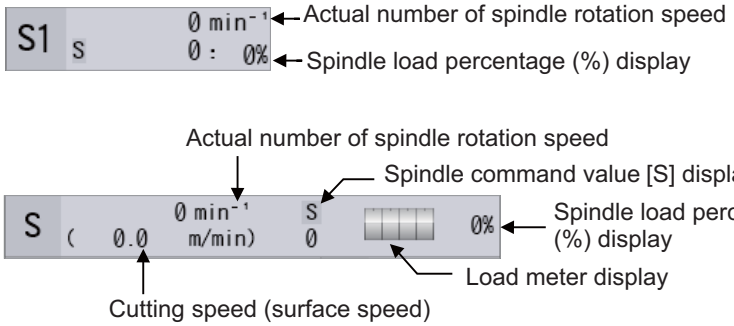
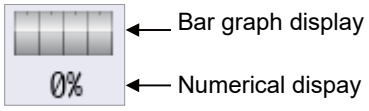
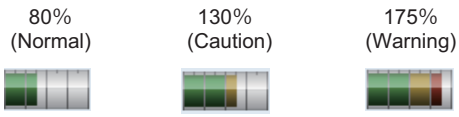
Note

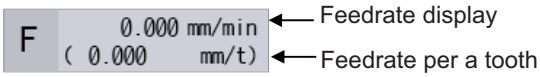
(1) The maximum number of axes that can be displayed is 5 axes. Press the menu [Next axis] to switch display to display the subsequent axes.

Display items

	Display items	Details
(1)	Counter display	This area displays the counter of the relative position or workpiece coordinate position. If each axis is at a specific position or in a specific status, the following status abbreviations are displayed. #1 to #8: 1st to 8th reference position ♦][: Servo OFF state ♦MR: Mirror image ♦><: Axis removed state ♦CT: Auxiliary axis state (Displayed when an axis that can be switched between the NC axis and auxiliary axis is set to the auxiliary axis state.) You can designate whether to consider the tool length compensation or tool radius compensation depending on the value of the parameter "#1287 ext23/bit4". The axis counter can be displayed or hidden depending on the value of the parameter "#1069 no_dsp". Also, if the axis is set to the diameter axis or uncontrol axis, the following symbol is displayed on the right of the axis name. *: Uncontrol axis (axis where any part system does not have control rights) φ: Diameter axis (When the target axis is equivalent to both the uncontrol axis and diameter axis, the symbol of the uncontrol axis is displayed.) <Note> ♦For PLC axis position counter, the status abbreviations, diameter axis and uncontrol axis are not displayed. ♦Refer to "Programming Manual" for details on the counter display for orbit machining.
(2)	Currently executed machining program (*1)	
	Main O10...	Displays the currently executed program number, sequence number, and block number.
	Sub O1234…	While a subprogram is executed, its program number, sequence number, and block number are displayed.
(3)	Buffer display	This displays the contents of the machining program currently being executed. The block being executed is highlighted. When the parameter "#1122 pglk_c" is enabled, the contents of the program are not displayed. Only "%" is displayed. When the parameter "#19431 Num display in prg" is set to "1", the variable, operation or function in the numerical part that follows each address is replaced with the calculated value and displayed for the current block and the next block. [Display example] Variable settings: #100=4., #101=5., #102=10. <When #19431 is set to "0">  <When #19431 is set to "1">  The variable or operation is replaced with the calculated value. <Note> ♦If the program comment includes the double-byte width character for one character code, the program may not be displayed correctly.

	Display items	Details
(4)	Tool No. display	This area displays the tool number, standby tool number, T command value, and tool diagram of the currently selected tool. <Monitor screen (Normal view) M system>  <Monitor screen (Normal view) L system>  <Monitor screen (Simple view) M system>  <Monitor screen (Simple view) L system>  (a) Tool No.: Displays the tool number of the currently selected tool. (b) Standby tool No.: Displayed only when standby tool 1 is designated. This is not displayed on the monitor screen (simple view). (c) Tool icon: Displays the type of the currently selected tool with an image. The icon designated on the tool management screen is displayed. (*2) If the lifetime of the tool in use exceeds 90% of the designated tool lifetime value, the background color of the tool icon is changed to yellow. When the lifetime of the tool in use is expired, the background color of the tool icon is changed to red, and it blinks. <Display example> Approaching tool life:  Tool life reached (background color flashing):  (d) Tool command value [T]: Displays the T command value that is commanded last. In the L system, when "#11038 T disp typ" is set to "1", the diameter is displayed, including the compensation number. (e) Diameter: Displays the diameter value of the currently selected tool. The value designated on the tool management screen is displayed. <Note> ♦For M system, if a tool with no diameter designated such as a cutting tool is selected, the diameter value is not displayed. For L system, if a tool with the diameter designated such as a milling tool is selected, the diameter is displayed.

	Display items	Details
(5)	Spindle rotation speed display	This area displays the cumulative number of spindle rotation speed as well as the spindle command value.  For L system, "0" is displayed in spindle cutting speed (surface speed). On the prototype or mass production screen, the values above vary depending on the value of the parameter "#1039 spinno".
(6)	Load meter display	This area displays the spindle load value (%) with a bar graph and numeric values. The simple view shows the load meter of up to two spindles.  The graph display color varies depending on the spindle load value. This item blinks when a warning occurs. The caution and warning thresholds conform to the parameters #3192 to #3194. <Display example> 

<table>
<tr><th colspan="2">Display items</th><th>Details</th></tr>
<tr><td>(7)</td><td>Speed display</td><td>This area displays feedrate.


Interpolation feeding: Displays the vector-direction speed (FA), dwell time, etc. of the axis currently being moved.
Each axis independent feeding: Displays the speed of the axis with the highest speed designated.
When the parameter "#1125 real_f" is set to "1", the actual machine feedrate is displayed.
The display units are as shown in the table below.
<table>
<tr><th>Metric/Inch</th><th colspan="2">Pattern</th><th>Display unit</th></tr>
<tr><td rowspan="8">Metric</td><td rowspan="2">G4 in progress</td><td>Feed per minute</td><td>S</td></tr>
<tr><td>Feed per revolution</td><td>rev</td></tr>
<tr><td rowspan="2">Thread cutting</td><td>Specifies the number of threads</td><td>mm/rev</td></tr>
<tr><td>Specifies thread read</td><td>mm/rev</td></tr>
<tr><td rowspan="2">Cutting feed</td><td>Rotary axis independent command</td><td>deg/min</td></tr>
<tr><td>Feed per revolution</td><td>mm/rev</td></tr>
<tr><td colspan="2">Any other than the above</td><td>mm/min</td></tr>
<tr><td rowspan="8">Inch</td><td rowspan="2">G4 in progress</td><td>Feed per minute</td><td>S</td></tr>
<tr><td>Feed per revolution</td><td>rev</td></tr>
<tr><td rowspan="2">Thread cutting</td><td>Specifies the number of threads</td><td>inch/rev</td></tr>
<tr><td>Specifies thread read</td><td>inch/rev</td></tr>
<tr><td rowspan="2">Cutting feed</td><td>Rotary axis independent command</td><td>deg/min</td></tr>
<tr><td>Feed per revolution</td><td>inch/rev</td></tr>
<tr><td colspan="2">Any other than the above</td><td>inch/min</td></tr>
</table></td></tr>
<tr><td>(8)</td><td>Cycle time display</td><td>This area displays the automatic operation time and cycle time.
The display item can be switched in the integrated time window.
•DAT: Date
•TIM: Time
•PON: Power ON
•OP: Automatic operation
•STL: Automatic start
•EX1: Ext time1
•EX2: Ext time2
•CYC: Cycle time</td></tr>
</table>

	Display items	Details
(9)	Control mode display	This area displays the NC control mode. •ISM: Inclined surface machining •HTP: High-speed synchronous tapping •CPC: Tool cutting point control (*3) •TCP: Tool center point control •MOR: Manual arbitrary reverse run mode •MC1 to MC9: Machining condition selection (*4) •CLC: Cutting load control •HA1: High-speed high-accuracy control II (G5P10000) •HA2: High-speed high-accuracy control III (G5P20000) •SSS: SSS control •CSC: Corner smooth control •VAC: Variable-acceleration pre-interpolation acceleration/deceleration •EAC: Allowable acceleration for each axis (*5) •SFR: Smooth fairing •FRG: Fairing •SXF: Selection of axis for feedrate command Up to six control modes are displayed beginning at the top.
(10)	G modal simple display	This area displays the following modal status. Display format varies depending on the status of G code group 24. •G command modal status of G code group 1 •G command modal status of G code group 3 •Currently selected workpiece coordinate system offset •Tool radius compensation modal, compensation number, tool shape amount, and tool radius wear •Tool length compensation modal, compensation axis name, compensation number, tool shape amount, and tool length wear
(11)	M and B commands	This area displays the command value for each of the M (miscellaneous function command value) and B (2nd miscellaneous function command value) functions. <In 1-part system display> The M (miscellaneous function command value) and B (2nd miscellaneous function command value) displays vary depending on the values of the parameters "#12005 Mfig" and "#1170 M2name".
(12)	M, S, T, and B commands (2-part system/3-part simultaneous display)	Using the menu [MST Chg] allows you to switch the MSTB display in the area to be operated. (For details, refer to "Changing MSTB display" in "10.2.5 Operation of 2-part/3-part/4-part System Simultaneous Display".)
(13)	M, T, and B commands (15-type/19-type) (2-part system/3-part simultaneous display)	This area displays the command value for each of the M (miscellaneous function command value), T, and B (2nd miscellaneous function command value) functions. The M (miscellaneous function command value) and B (2nd miscellaneous function command value) displays vary depending on the values of the parameters "#12005 Mfig" and "#1170 M2name".
(14)	Spindle rotation speed display (2-part system/3-part system/4-part system simultaneous display)	This area displays the spindle rotation speed and the spindle command value that are designated in the parameters "#8925 to #8928" ("SP on 1st part sys" to "SP on 4th part sys") and "#8958 to #8961" ("SP on 5th part sys" to "SP on 8th part sys").

<table>
<tr><th>Display items</th><th>Details</th></tr>
<tr><td>(15) Workpiece machining count display</td><td>This area displays the current number of workpiece machining.
The display depends on the setting of the parameter #8001 to #8003.
However, when the parameter "#8001 WRK COUNT M" is set to "0", the number of workpiece machining is not counted, and it is not displayed on the screen.
The setting value display of the parameter #8001 is changed depending on the combination with the setting value of the parameter "#1759 cfgPR09/bit5" (Increase display digits of M code for counting workpiece machining).
[Display example]
<table>
<tr><th>#8001</th><th>#1759 cfgPR09/bit5</th><th>Screen display</th></tr>
<tr><td>330</td><td>0 (The first two digits)</td><td>
</td></tr>
<tr><td>330</td><td>1 (Three digits)</td><td>
</td></tr>
</table>
<Note>
♦When the following M codes are set in #8001, the number of workpiece machining is not counted.
- M codes used for M code macro
- M98, M99, M198
- M96, M97</td></tr>
<tr><td>(16) Selective display area</td><td>This area displays NC data that is selected in the parameter "#8940 Set select display".
Depending on the selected NC data, you can press the [PAGE↑], [PAGE↓], [↑], [↓], [←], [→], [→|], or [|←] key to scroll pages.
Pressing the [INPUT] key displays the window that allows you to operate NC data displayed in the selective display area.
The selective display area can be displayed on the following screens:
♦8.4-type/10.4-type: Normal view
♦15-type/19-type: Single counter / Dual counter / Quad counter
For 8.4-type/10.4-type display units, this option can be activated by setting parameter "#8973 Selectable display" to "1".
The screen shown in the selective display area is selected using the "#8940 Set select display" parameter.</td></tr>
<tr><td>(17) Rate display (15-type/19-type)</td><td>The upper side displays the execution position in the main program in percentage (rate) when the entire size of the currently executed main program is assumed to be 100%. The lower side displays the execution position in the sub program in percentage (rate) when the entire size of the currently executed sub program is assumed to be 100%.</td></tr>
</table>

(*1) If the program number (program name) exceeds the number of digits that can be displayed, the last digit is indicated by an asterisk (*).
The device name displays the device name detected by operation search. However, "232C" is displayed at computer link B even if no operation search is carried out.
If the sequence number exceeds six digits, only the low-order six digits are displayed.

(*2) If the data of the designated tool number has not been registered on the tool management screen, the tool icon is not displayed.

(*3) When tool cutting point control is enabled, tool center point control is always enabled. Thus, "TCP" is not displayed while the control mode "CPC" is being displayed.

(*4) When selecting machining conditions, the following is displayed according to G command or the selected application and condition.

	Use	Condition	G command
MC1	Machining application 1	Condition 1	G120.1 P1 Q1
MC2	Machining application 1	Condition 2	G120.1 P1 Q2
MC3	Machining application 1	Condition 3	G120.1 P1 Q3
MC4	Machining application 2	Condition 1	G120.1 P2 Q1
MC5	Machining application 2	Condition 2	G120.1 P2 Q2

	Use	Condition	G command
MC6	Machining application 2	Condition 3	G120.1 P2 Q3
MC7	Machining application 3	Condition 1	G120.1 P3 Q1
MC8	Machining application 3	Condition 2	G120.1 P3 Q2
MC9	Machining application 3	Condition 3	G120.1 P3 Q3

(*5) When the variable-acceleration pre-interpolation acceleration/deceleration is enabled, the allowable acceleration speed for each axis is always enabled, and "EAC" is not displayed while the control mode "VAC" is being displayed.

Sub-menus displayed by pressing the menu [Dsp Chg]

Sub-menus displayed by pressing the menu [Dsp Chg], when 2-part system simultaneous view is invalid (#11019=0) with 8.4-type or 10.4-type.

Menus	Details
Normal view	Switches the display format to the normal view.
Simple view	Switches the display format to the simple view.
Counter #1	Switches counter type 1. (*1)
Counter #2	Switches counter type 2. (*1) (*2)
Counter #3	Switches counter type 3. (*1)
Counter #4	Switches counter type 4. (*1) (*2)
Counter #5	Switches counter type 5. (*3)

(*1) This menu cannot be operated in simple view mode.

(*2) This menu cannot be operated in normal view mode (6 axes or more).

(*3) This menu cannot be operated in normal view mode.

Note

(1) The selected display format is also held when the power is turned on again.

Sub-menus displayed by pressing the menu [Dsp Chg], when 2-part system simultaneous view is invalid (#11019=0) with 15-type or 19-type

Menus	Details
Single counter	Switches the display format to single counter.
Dual counter	Switches the display format to dual counter.
Quad counter	Switches the display format to quad counter.
Counter #1	Switches counter type 1. (*1)
Counter #2	Switches counter type 2. (*1) (*2)
Counter #3	Switches counter type 3. (*1)
Counter #4	Switches counter type 4. (*1) (*2)
Counter #5	Switches counter type 5. (*2) (*3)

(*1) This menu cannot be operated in single counter mode.

(*2) This menu cannot be operated in dual counter mode.

(*3) This menu cannot be operated in quad counter mode.

Note

(1) The selected display format is also held when the power is turned on again.

Sub-menus displayed by pressing the menu [Dsp Chg], when 2-part/3-part/4-part system simultaneous display is valid (#11019 = 1 to 4)

Menus	Details
Counter expand	Switches the counter display format to counter1 (expand).
Dual counter	Switches the counter display format to dual counter.
Quad counter	Switches the counter display format to quad counter.
MSTB view	Switches the MSTB display for the area to be operated. This menu cannot be operated for 2-part or 4-part system simultaneous display on 15-type/19-type display unit.
Counter #1	Switches counter type 1.
Counter #2	Switches counter type 2. (*1) (*2)
Counter #3	Switches counter type 3. (*1) (*3)
Counter #4	Switches counter type 4. (*1) (*3)

(*1) This menu cannot be operated in single counter mode.

(*2) This menu may not be able to be operated in dual counter mode depending on the number of displayed axes.

(*3) This menu cannot be operated in dual counter mode.

Note

(1) The selected display format is also held when the power is turned on again.

10.2.2 Switching the Display Format

Monitr – 

The monitor screen display format can be switched using the menu [Dsp Chg]. The switchable display format varies depending on the value of the parameter "#11019 2-system display".

(1) In the 1-part system display (#11019 = 0), switch the display format between the normal view and simple view or among single counter, dual counter, and quad counter.

(2) In the 2-part/3-part/4-part system simultaneous display (#11019 = 1 to 4), switch the display format among the counter1 (expand), dual counter and quad counter.

Switching the normal view and simple view

When the parameter "#11019 2-system display" (2-part system simultaneous display) is disabled, you can press the sub menu [Normal view] or [Simple view] in the [Dsp Chg] to switch the screen to the normal view or simple view. In 15-type/19-type, the screen can be switched using the sub menu [Single counter], [Dual counter], or [Quad counter].

Switching the Counter Display Format

When the parameter "#11019 2-system display" (2-part system simultaneous display) is enabled, the counter display format can be switched to the following three modes.

- Counter1 (expand): Displays one counter (Counter type 1) for each part system.
- Dual counter: Displays two counters (Counter types 1 and 2) for each part system.
- Quad counter: Displays four counters (Counter types 1 to 4) for each part system.

Note

(1) For details on the screen image, refer to Screens 9 to 19 in "10.2.1 Screen Layout".
You can press the sub-menu [Counterexpand], [Dual counter], or [Quad counter] in the menu [Dsp Chg] to switch the display format.

10.2.3 Switching the Counter Types

Monitr – Dsp.Chg

The counter mode on the monitor screen (monitor screen (normal view), monitor screen (simple view), and 2-part/3-part/4-part system simultaneous display screen) can be switched using the parameter or the menu [Counter].

(1) Switching the counter type in the parameter setting

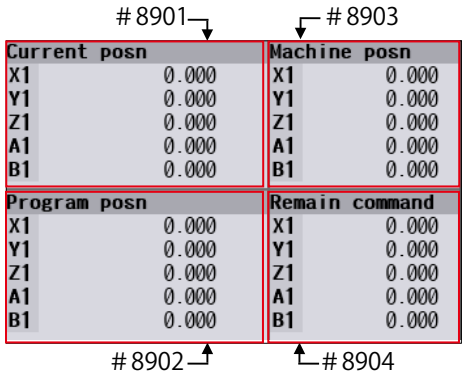
(2) Switching the counter type in the menu [Counter]

Switching in the parameter setting

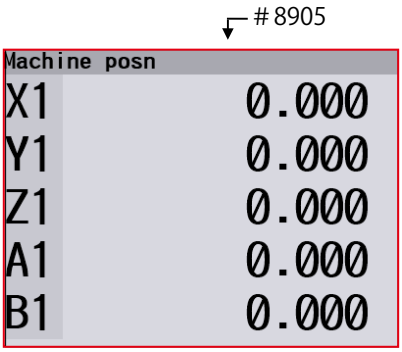
The type of the counter to be displayed on the monitor screen can be switched using each of the counter type parameters [#8901 Counter type 1] to [#8905 Counter type 5]. The figure below shows the relationship between counters and parameters.

■1-part system display

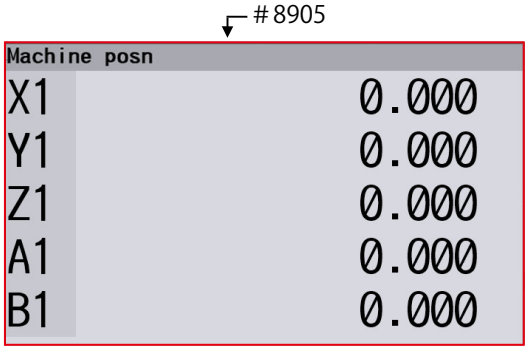
<Normal view>



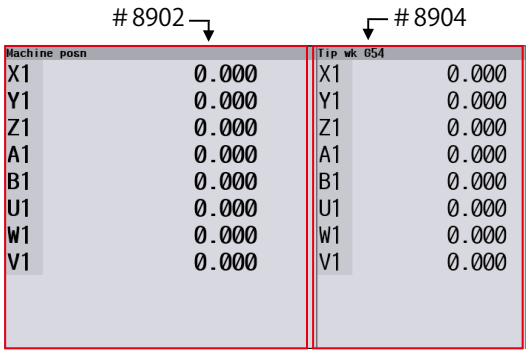
<Simple view>



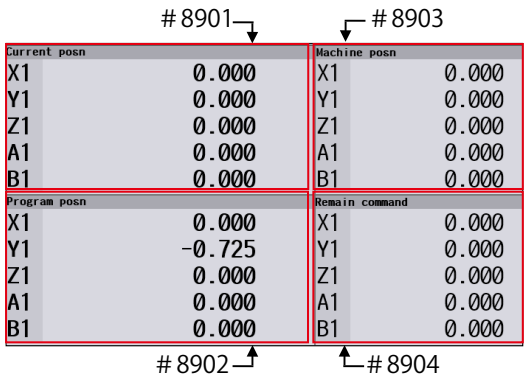
<15-type/19-type, single counter>



<15-type/19-type, dual counter>



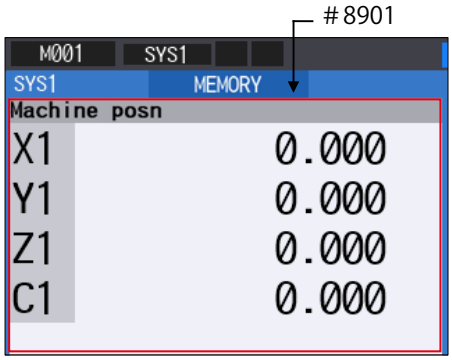
<15-type/19-type, quad counter>



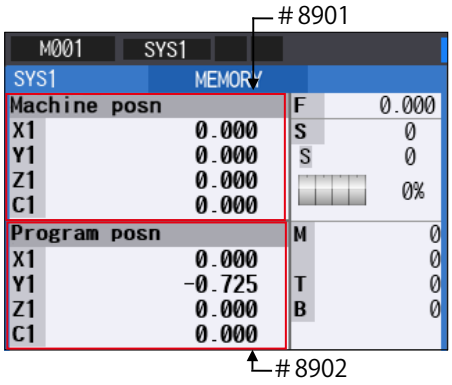
■2-part system/3-part system/4-part system simultaneous display

The following counter type is displayed in the target operation area on the left.
Also, the same counter type is displayed in the target operation area on the right.

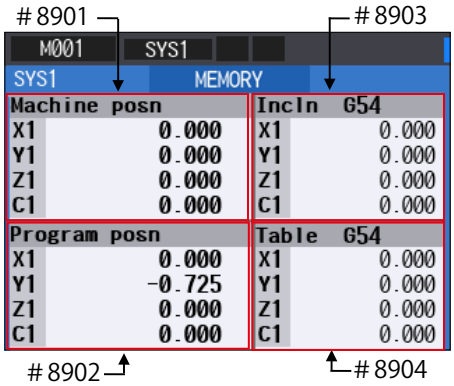
<Counterexpand>



<Dual counter>



<Quad counter (2-part/3-part system simultaneous display)>



<Quad counter (4-part system simultaneous display)>

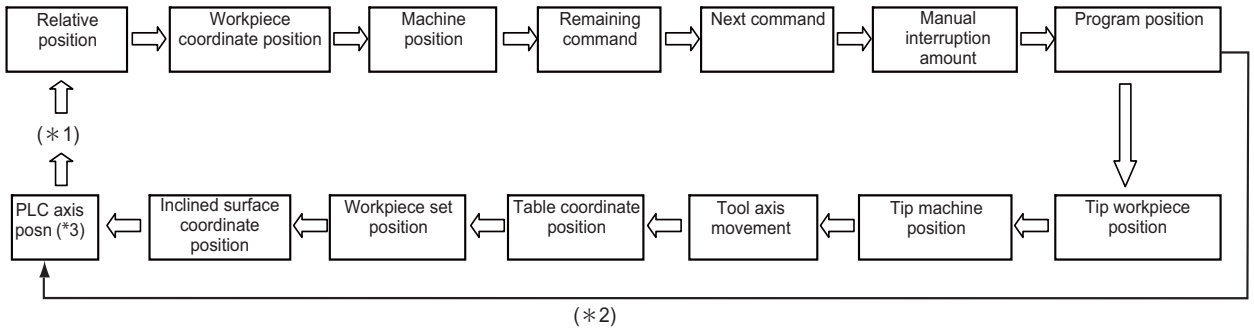


Switching the counter type using the menu

The type of the counter to be displayed on the monitor screen can be changed using the sub-menu [Counter] under the menu [Dsp sw.]. If the counter type is switched using the menus [Counter #1] to [Counter #5], the values of the parameters "#8901 Counter type 1" to "8905 Counter type 5" are rewritten.

However, when data protection is valid, the displayed counter type can be switched using the menu, but the value of each parameter cannot be rewritten. In this case, if the display returns to the monitor screen after switching to another screen while the displayed counter type is different from the parameter value, the displayed counter is set to the counter type that matches the parameter value.

The figure below shows the counter types that can be displayed using the menu [Counter] as well as the switching order.



(*1) If a counter other than the 14 types above ("Relative position" to "Inclined surface coordinate position") is set to the parameter ("#8901 Counter type 1" to "#8905 Counter type 5") before the counter type is switched using [Switch counter], the counter type is switched among 15 types of counters, including that counter. When the counter type is switched to a counter type other than the 14 counter types above, it is held until the power is turned OFF.
Furthermore, if the 14 counter types above are designated in parameters before the counter type is switched using the menu, the counter type is switched among these 14 counter types.

(*2) When any of the following specifications is enabled, "Tip workpiece position", "Tip machine position", "Tool axis movement", "Table coordinate position", "Workpiece set position" and "Inclined surface coordinate position" counters can be switched.

- Tool center point control
- Tool length compensation along the tool axis
- Tool handle feed & interruption
- Workpiece installation error compensation
- Inclined surface machining command
- Simple inclined surface machining command
- 3-dimensional tool radius compensation (tool's vertical-direction compensation)
- 3-dimensional manual feed
- R-Navi
- Program format switch

(*3) When the parameter "#11091 PLC counter valid" is enabled, it switches to [PLC Axis] counter. [PLC Axis] counter is common for part systems. Even when the displayed part system is switched, the value of [PLC Axis] is not changed. When the axis name of PLC axis is not set, the axis name of PLC axis counter is displayed as "P + axis No." (P1, P2, ...). The axis set to non-display axis by the parameter "#1069 no_dsp" is not displayed.

The axis set by any of the following parameters is not displayed for switching.

- #1280 ext16/bit2 (Interchange coordinate position display)
- #1494 dsp_ax_change (Switching the counter display axis order)
- #1537 crsax[1] to #1544 crsax[8] (Axis interchange by mixed control function)

The axis set as the auxiliary axis by the parameter "#12800 chgauxno" is displayed in PLC axis counter.

10.2.4 Switching the Spindle No. to be Displayed on Counter

Monitr – 

The number of counter axes that can be displayed on the monitor screen is as follows. To refer to the subsequent axes, select the menu [Next axis].

- Monitor screen in 1-part system view (normal view): Up to 10 axes
- Monitor screen in 1-part system view (simple view): Up to 5 axes
- 2-part system simultaneous view (Counter1 (expand)): Up to 4 axes
- 2-part system simultaneous view (Dual counter): Up to 8 axes
- 2-part system simultaneous view (Quad counter): Up to 4 axes
- 3-part system simultaneous view (Counter1 (expand)): Up to 5 axes
- 3-part system simultaneous view (Dual counter): Up to 10 axes
- 3-part system simultaneous view (Dual counter): Up to 5 axes

Note

(1) When the mixed control (cross axis control) or arbitrary axis exchange control is enabled, the menu [Next axis] is displayed even if there is no axis to be switched.

10.2.5 Operation of 2-part/3-part/4-part System Simultaneous Display

Switching the area to be operated (right/left)

The area to be operated (right/left) can be switched by touching in the area using the tab key (|← →|) or on the touchscreen.

When the area to be operated is switched, the part system name to be operated is displayed on the selection bar. At NC startup, the left side is set as the area to be operated.

Note

(1) The area to be operated cannot be switched while the pop-up window is being displayed.
(2) The area to be operated can be switched while the software keyboard is being displayed.

Switching the display part system No.

The part system is switched for the area to be operated (right/left).

When the part system displayed in the non-target area is designated by incrementing displayed part systems by one each time you press the $←→$ key, the area to be operated is switched.

If the available number of displayed part systems is reached, the first part system is displayed.

Note

(1) The same operation is also applied when you click a part system name in the selection bar on the touchscreen.
(2) If the base common parameter "#11035 Sys. change limit" is set to "1", it disables part systems from being switched using the $←→$ key or on the touchscreen.

Changing MSTB display

On the monitor screen in 2-part/3-part system simultaneous view, you can press the [MST Chg] main menu to change the MSTB display for the area to be operated.

The MSTB display type is held for each part system when the screen is switched or the power is turned ON again.

<2-part system simultaneous view: Counter1 (expand) / Quad counter>

There are the following two types of MSTB displays, which are toggled each time the menu [MST Chg] is pressed.

- MSTB display type (*1) (*2) (*3)
- MSTB + Workpiece machining count display type (*1) (*2)

(*1) The S command is used to designate the spindle to be displayed for each part system using parameters (#8925 to #8928 ("SP on 1st part sys" to "SP on 4th part sys") and #8958 to #8961 ("SP on 5th part sys" to "SP on 8th part sys")). Up to two spindles can be designated in each part system. The M (miscellaneous function command value), T (tool command value), and B (2nd miscellaneous function command value) functions are displayed one by one regardless of the value of the parameter "#12005 Mfig" (M count), "#12009 Tfig" (T count), or "#12011 Bfig" (B count). However, the B (2nd miscellaneous function command value) function is displayed or hidden depending on the value of the parameter "#1170 M2name" (2nd miscellaneous code).

(*2) The menu [MST Chg] is not available in the following cases:

- The parameter "#8001 WRK COUNT M" is set to "0".
- "F" is set to the low-order byte of parameters #8925 to #8928 ("SP on 1st part sys" to "SP on 4th part sys") and #8958 to #8961 ("SP on 5th part sys" to "SP on 8th part sys"), or the parameter "#1039 spinno" is set to "1".

(*3) When the number of spindles is set to "1" or only one spindle is displayed depending on the operation parameters (#8925 to #8928 ("SP on 1st part sys" to "SP on 4th part sys") and #8958 to #8961 ("SP on 5th part sys" to "SP on 8th part sys"), the load meter of the target spindle is displayed at the position at which the second spindle is to be displayed.

<2-part/3-part system simultaneous display: Dual counter>

There are the following six types of MSTB displays, which are toggled each time the menu [MST Chg] is pressed.

- MSTB display type (*1)
- S display type 1
- S display type 2
- MTB display type (2-part system simultaneous display only)
- MTB + Workpiece machining count display type
- MSTB + Workpiece machining count display type (*1)

(*1) When the number of spindles is set to "1" or only one spindle is displayed depending on the operation parameters (#8925 to #8928 ("SP on 1st part sys" to "SP on 4th part sys") and #8958 to #8961 ("SP on 5th part sys" to "SP on 8th part sys"), the load meter of the target spindle is displayed at the position at which the second spindle is to be displayed.

Customizing the screen display (Select display area (parameter "#8973 Selective display" = "1"))

A part of the monitor screen (normal view) can be customized to suit user's need (to display data which a user always needs to refer to). The select display area shows the screen that is selected using the parameter "#8940 Set select display".

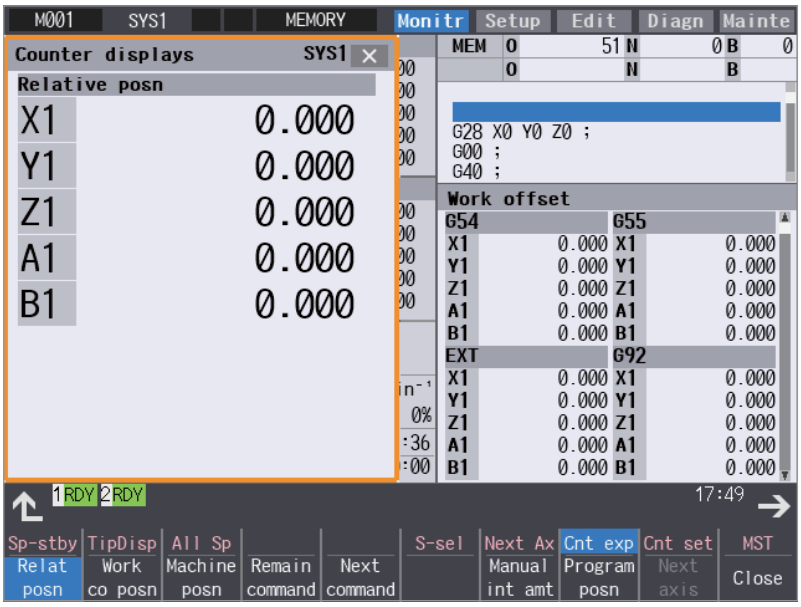
Customizing the displaying order of the part systems

When multiple part systems (2-, 3-, or 4-part system) are displayed simultaneously, they are displayed in ascending order with 1-part system on the leftmost area after the power is turned ON.
You can customize the displaying order of part systems by setting parameter "11791 Disp area1 sysno" to "11794 Disp area4 sysno".

10.2.6 Counter All-axis Display

Monitr – Cnt exp

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



Menus

Menus	Details
Relat posn	This displays the currently executed position.
Work co posn	This displays the G54 to G59 workpiece coordinate system modal No. and the workpiece coordinate position in that workpiece coordinate system.
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.
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.)
Next command	This displays the details of the command in the block executed after the block currently being executed.
Manual int amt	This displays the amount moved with the manual mode while the manual absolute switch was OFF. When the parameter "# 8173 Hold intr amount" is set to "0", interrupt amount is cleared by pressing the reset button.
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. When the parameter "# 8173 Hold intr amount" is set to "0", interrupt amount is cleared by pressing the reset button.
Close	This closes the pop-up window and quits this function.
Tip wk posn	This displays the position of the tool center from the workpiece coordinate reference position in the selected workpiece coordinate system.

Menus	Details
Tip machine	This displays the position of the tool center from the machine coordinate system reference position in the machine coordinate system.
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.
Table co posn	This displays the table coordinate.
Wk Comp co posn	This displays the workpiece installation position coordinate.
Incline co posn	This displays the inclined surface coordinate.
PLC Axis	This displays the machine position of PLC axis.
Next axis	This switches the axis to be displayed on the counter. This switches the axes to be displayed on the counter between "1st axis to 12th axis" and "13th axis or after".

Note

(1) The type of counter displayed first when the pop-up display appears is the relative position.
However, when any of the targeted specifications (*1) is valid, "Tip workpiece position" is displayed. The counter which appears next is the type selected previously.

(2) The menus [Tip wk posn], [Tip machine], [Pulse] and [Table co posn] appear when any of the targeted specifications (*1) is valid.

(3) The menu [Wk Comp co posn] is displayed when the workpiece installation error compensation specification is ON.

(4) The menu [Incline co posn] is displayed when any of the specifications (inclined surface machining command, simple inclined surface machining command or R-Navi) is valid.

(5) The menu [PLC Axis] is displayed when the parameter "#11091 PLC counter valid" is valid.
When the axis name of PLC axis is not set, the axis name of PLC axis counter is displayed as "P + axis No." (P1, P2, ...).
The axis set to the axis with no display in the parameter "#1069 no_dsp" is not displayed.

The axis set by any of the following parameters is not displayed for switching.

- #1280 ext16/bit2 (Interchange coordinate position display)
- #1494 dsp_ax_change (Switching the counter display axis order)
- #1537 crsax[1] to #1544 crsax[8] (Axis interchange by mixed control function)

The axis set as the auxiliary axis by the parameter "#12800 chgauxno" is displayed in PLC axis counter.

(*1) Targeted specifications

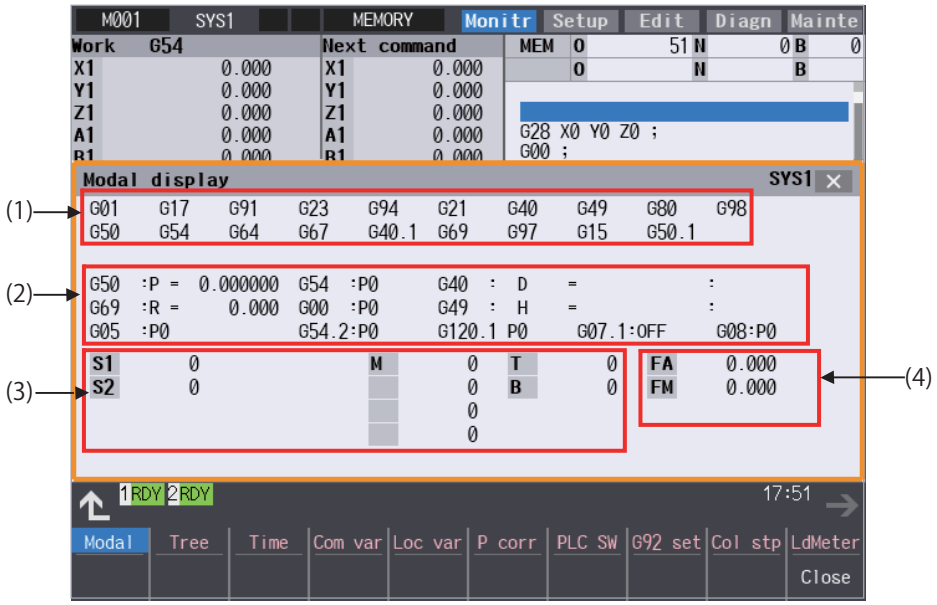
- Tool center point control
- Tool length compensation along the tool axis
- Tool handle feed & interruption
- Workpiece installation error compensation
- Inclined surface machining command
- Simple inclined surface machining command
- 3-dimensional tool radius compensation (tool's vertical-direction compensation)
- 3-dimensional manual feed
- R-Navi

10.2.7 Modal Display

Monitr – Modal

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

<M system>



Even if operating area is switched in the 2-part system simultaneous display, the position of the pop-up window will not change. It is always displayed at the bottom of the screen.

Display items

Display items	Details	
(1) G modal list	The list below shows the G command modal status of the currently executed groups 1 to 20 or group 24. Each item displays the following information. G00: Displays the G command modal status of the currently executed group 1. G17: Displays the G command modal status of the currently executed group 2. : Displays the G command modal status of each group in sequence. G50.1: Displays the G command modal status of the currently executed group 19.	
(2) Details of G modal	This item displays detailed information of the G command modal status of the currently executed group. Each item displays the following information.	
	G50 :P= 0.000000	Scaling factor
	G54.1:P1	Workpiece coordinate system offset No.
	G42 : D 10= 0.000000: 0.000000 G42: D10: = 0.000000: 0.000000	 Tool radius compensation modal Offset num Shape compensation amount of tool radius Tool radius wear amount
	G69 :R = 0.000	Coordinate rotation angle (Displayed in the range of ±360°.)
	G54.4:P1	Workpiece installation error compensation No.

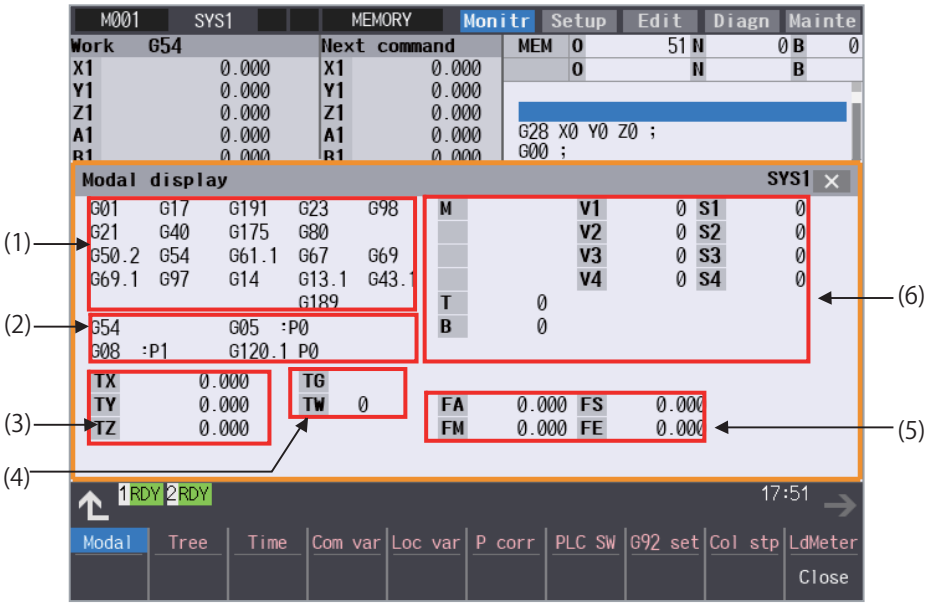
Display items	Details	
(2) Details of G modal (continued)	G43 :ZH 50= 0.000000: 0.000000	
	G43:	Tool length compensation modal
	Z	Compensation axis name
	H50	Offset num
	= 0.000000:	Offset
	0.000000	Tool length wear amount
	G43 :ZH 50= 0.000000: 0.000000	
	G43:	Tool length compensation modal
	Z	Compensation axis name
	H50	Offset num
	= 0.000000:	Offset
	0.000000	Tool length wear amount
	G05.1:P1000	
	(Display example)	
	G05 :P0	High-speed machining mode
	G05.1: Q1	High-speed high-accuracy control I
	G05.1: Q2	Spline interpolation
	G05 :P10000	High-speed high-accuracy control II (SSS control OFF)
	G05 :P1	High-speed machining mode I
	G05 :P2	High-speed machining mode II
	G54.2 P0	Workpiece position offset for rotary axis
	VAC	Variable-acceleration pre-interpolation acceleration/deceleration enabled
	CSC	Corner smooth control enabled
	SFR	Smooth fairing ON
	SSS (*1)	SSS enabled
	G120.1 P1 Q1	Machining Condition Selection I
	(Display example)	
	G120.1 P0	Reference parameter selected
	G120.1 P1 Q2 (*2)	Each machining condition parameter selected Pn: (n=1 to 3) Machining purpose Qm: (m=1 to 3) Condition
	G07.1: ON G07.1: OFF	Cylindrical interpolation modal (Fixed to G07.1) Cylindrical interpolation mode (ON/OFF)
	G08: P0 G08: P1	High-accuracy control mode OFF High-accuracy control mode ON
(3) MSTB display area	This area displays the program command modal value for each of the currently executed commands. Each item displays the following information. ♦S command ♦M command ♦T command ♦2nd miscellaneous function (B) command The number of S displays varies depending on the value of the parameter "#1039 spinno" (Number of spindles). The number of M displays varies depending on the value of the parameter "#12005 Mfig" (M count). The number of T or B displays is set to "1" regardless of the value of the parameters "#12009 Tfig" (T count) and "#12011 Bfig" (B count). However, for the B display, the value is displayed or hidden depending on the value of the parameter (#1170 M2name 2nd miscellaneous code). For the T display, only the tool number is displayed regardless of the value of the parameter "#11038 T disp type".	

Display items	Details
(4) F modal display area	This area displays the F modal value and manual feedrate. FA: Displays the program command F modal value of the currently executed command. FM: Displays the manual feedrate.

(*1) Displays the current control mode. When multiple control modes are enabled simultaneously, one of them is displayed in the order in which the priority is higher. (The priority is higher beginning at the top.)

(*2) G121 command: Machining Condition Selection I Cancel
Changes to the machining condition selection command that is selected in [Setup] - [MacCond] screen.

<L system>



Even if operating area is switched in the 2-part system simultaneous display, the position of the pop-up window will not change. It is always displayed at the bottom of the screen.

Display items

Display items	Details
(1) G modal list	The list below shows the G command modal status of the currently executed groups 1 to 20 or group 24 (*1). Each item displays the following information. G00: Displays the G command modal status of the currently executed group 1. G17: Displays the G command modal status of the currently executed group 2. : Displays the G command modal status of each group in sequence. G189: Displays the G command modal status of the currently executed group 24.

Display items	Details	
(2) Details of G modal	This item displays detailed information of the G command modal status of the currently executed group. Each item displays the following information.	
	G54.1:P10	Workpiece coordinate system offset No.
	The data format of the workpiece coordinate system offset number is as follows:	
	G5x :P (x = 4 to 9)	G54 Workpiece coordinate system selection 1 G55 Workpiece coordinate system selection 2 G56 Workpiece coordinate system selection 3 G57 Workpiece coordinate system selection 4 G58 Workpiece coordinate system selection 5 G59 Workpiece coordinate system selection 6
	G54.1: Pyy (yy = 1 to 96)	G54.1 Workpiece coordinate system selection The number of extension sets is assigned to 0, 48, or 96 sets depending on the optional specifications.
	When the high-speed machining mode, high-speed high-accuracy control, or direct command mode specification is enabled, the status of the high-speed machining mode, high-speed high-accuracy control, or direct command mode is displayed.	
	G05 :P0	OFF
	G05 :P1	High-speed machining mode I ON
	G05 :P2	High-speed machining mode II ON
	G05.1: Q0	High-speed high-accuracy control I OFF
	G05.1: Q1	High-speed high-accuracy control I ON
	G05 :P10000	High-speed high-accuracy control II ON
	G05 P4	Direct command mode ON
	Displays the current control mode. When multiple control modes are enabled simultaneously, one of them is displayed in the order in which the priority is higher. (The priority is higher beginning at the top.)	
	VAC	Variable-acceleration pre-interpolation acceleration/deceleration enabled
	CSC	Corner smooth control enabled
	SFR	Smooth fairing ON
	SSS	SSS enabled
	When the high-accuracy control specification is enabled, the status of the high-accuracy control mode is displayed.	
	G08:P0	High-accuracy control mode OFF
	G08:P1	High-accuracy control mode ON
	Displays the currently selected machining condition parameter group. * If the reference parameter is selected, "G120.1 P0" is displayed.	
	G120.1 P1 Q2	Machining Condition Selection I
	(Display example)	
	G120.1 P0	Reference parameter selected
	G120.1 P1 Q1 (*2)	Each machining condition parameter selected Pn: (n=1 to 3) Machining purpose Qm: (m=1 to 3) Condition
(3) Tool length and wear compensation amount display area	This area displays the tool length and wear compensation amount. TX: Displays the sum of the tool length and wear compensation amount of the first axis for the tool in use. (*3) TZ: Displays the sum of the tool length and wear compensation amount of the second axis for the tool in use. (*3) TY: Displays the sum of the tool length and wear compensation amount of the additional axis for the tool in use. (*4) TC: Displays the sum of the tool length and wear compensation amount of the second additional axis for the tool in use. (*5)	

Display items	Details
(4) Compensation number display area	This area displays the compensation number. TG: Displays the tool length compensation number. TW: Displays the wear compensation number.
(5) F modal area	This area displays the F modal value and manual feedrate. FA: Displays the program command asynchronous feedrate modal value of the currently executed command. FM: Displays the manual feedrate. FS: Displays the program command synchronous feedrate modal value of the currently executed command. FE: Displays the thread lead command synchronous feedrate modal value of the currently executed command.
(6) MSTB area and constant surface speed control (constant surface speed S) area	<MSTB area> This area displays the program command modal value for each of the currently executed commands. Each item displays the following information: •S command •M command •T command •2nd miscellaneous function (B) command The number of S displays varies depending on the value of the parameter "#1039 spinno" (Number of spindles). The number of M displays varies depending on the value of the parameter "#12005 Mfig" (M count). The number of T or B displays is set to "1" regardless of the value of the parameter "#12009 Tfig" (T count) and "#12011 Bfig" (B count). However, for the B display, the value is displayed or hidden depending on the value of the parameter (#1170 M2name 2nd miscellaneous code). For the T display, only the tool number is displayed regardless of the value of the parameter "#11038 T disp type".

(*1) The status of group 24 command modal is displayed only when the program format switch specification is valid and the parameter "#1254 set26/bit1" is set to "1".

(*2) G121 command: Machining Condition Selection I Cancel
Changes to the machining condition selection command that is selected in [Setup] - [MacCond] screen.

(*3) Displays the axis name that is set to the first and second axes in the part system for the base axis specification parameter "#1013 axname". (If no axis name is set to "#1013 axname", the axis name item is blank.)

(*4) When the parameter "#1281 ext17 bit1" (tool offset additional axis selection) is set to "0", the display will change as follows by the parameter "#1520 Tchg34" (additional axis tool) setting.
#1520 Tchg34 = 0: Displays the axis name configured as the third axis in the part system by the parameter "1013 axname".
#1520 Tchg34 = 1: Displays the axis name configured as the 4th axis in the part system.
(If no axis name is set to "#1013 axname", the axis name item is blank.)
When the parameter "#1281 ext17 bit1" (tool offset additional axis selection) is set to "1", the name of the axis designated by the parameter "#1027 base_J" (basic axis J) is displayed. (If the axis name designated in "#1027 base_J" is not set to "#1013 axname", the axis name item is blank.)
In the two-axis configuration system, the third row is not displayed.

(*5) The axis name which is designated by the parameter "#12104 2nd add T-ofs set" (2nd additional axis tool offset setting) is displayed. The 4th line is not displayed when the parameter "#12103 2nd add T-ofs ON" (2nd additional axis tool offset ON) is not configured.

Menus

Menus	Details
Close	This closes the pop-up window and quits this function.

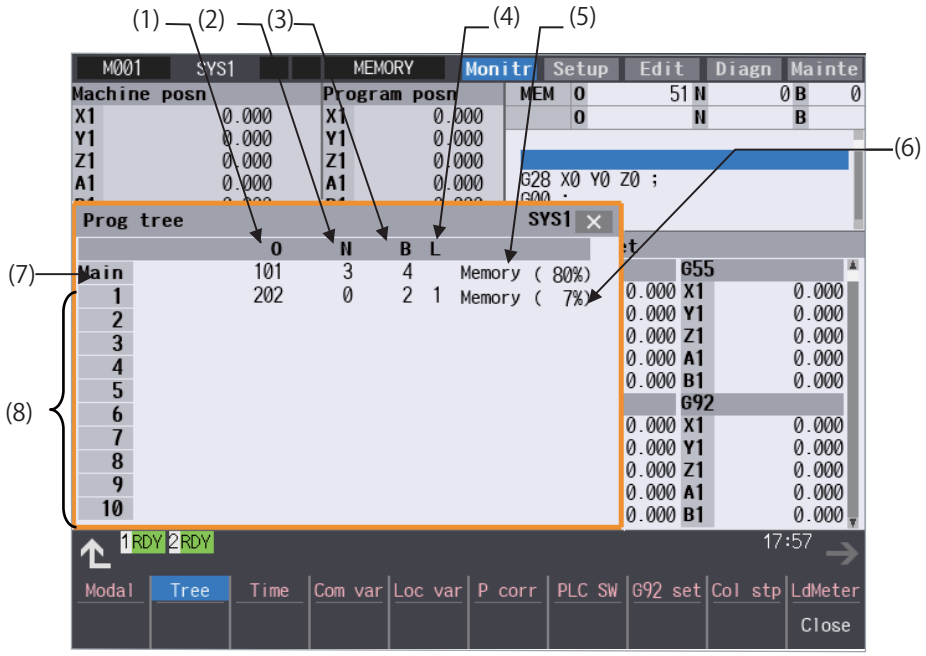
Note

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

10.2.8 Program Tree Display

Monitr – Tree

This displays the main program, subprogram, MDI interrupt and user macro call nesting structure.
This also displays the execution position of the main program and the subprogram (the lowest level of the nest only) as a percentage during the automatic operation.



Display items

Display items		Details
(1)	O	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 99999999)
(3)	B	Block No. (0 to 99999)
(4)	L	Remaining number of subprogram repetitions (0 to 99)

Display items		Details
(5)	Devices	Operation device (0 to 7 characters) Displays the program storage device and MDI interruption. Memory: NC memory Memory2: NC memory 2 HD: Hard disk Memory card: Memory card DS: Data server RS-232C: NC Serial MDI: MDI interruption USB Memory: USB Memory
(6)	(%)	Program execution rate display Assuming the whole machining program size as 100%, the execution position during an automatic operation is displayed in % (percentage). When the automatic operation is not executed, 0% is displayed for the main program's percentage display. The subprogram's percentage is displayed only for the lowest level of the nest and other subprograms will display blank.
(7)	Main	Main program
(8)	1 to 10	Called subprogram, user macro call and MDI interrupt

Menus

Menus	Details
Close	This closes the pop-up window and quits this function.

Precautions

(1) Calling the subprogram in the subprogram (M98) is available up to 10 nest levels, in case the main program is counted as 0. If it exceeds the limit, a program error occurs.

(2) Calling the subprogram of user macro is available from level 1 to 4.

(3) Nesting level call is available up to 10 nest levels in total of subprogram call, user macro call and MDI interruption.

(4) Subprogram called by fixed cycle is not displayed.

10.2.9 Integrated Time Display

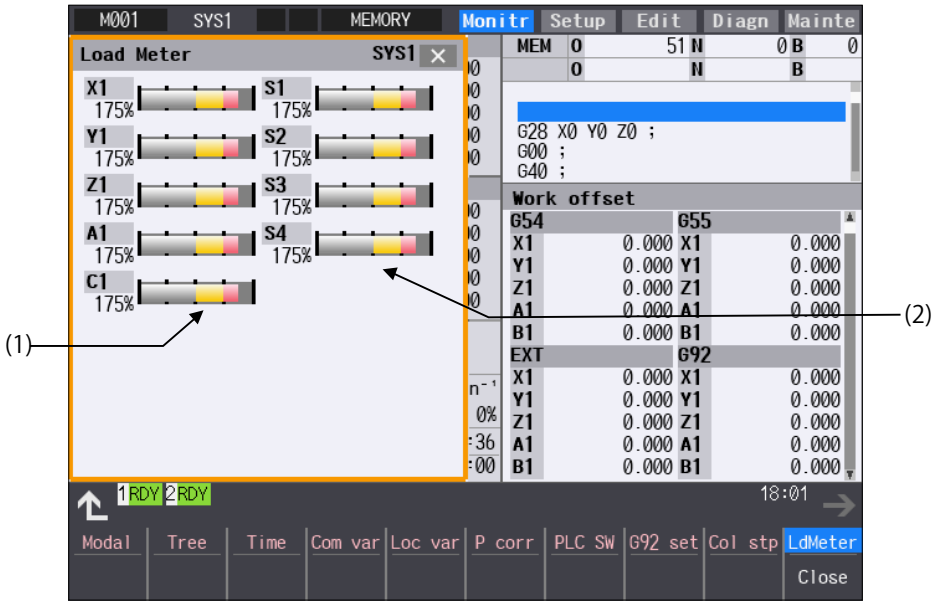


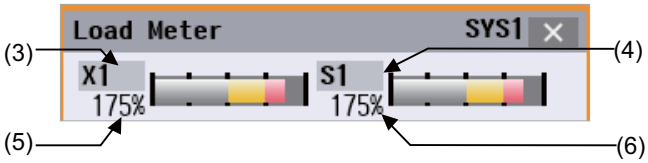
For details, refer to "9.3.1 Integrated Time Display".

10.2.10 Load Meter Display

Monitr – LdMeter

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





Display items

	Display items	Details
(1)	NC axis load meter	Up to eight NC axes loads (%) are displayed. The display color of graph is switched in accordance with the NC axis load. The threshold values of cautions (yellow) and warning (red) are determined according to the parameters (#2643 to #2645).
(2)	Spindle load meter	Up to eight spindles loads (%) are displayed. The graph display color varies depending on the spindle load value. The threshold values of cautions (yellow) and warning (red) are determined according to the parameters (#3192 to #3194).
(3)	NC axis name	This displays the axis name set with the parameter "#1022 axname2".
(4)	Spindle name	This displays "S + Spindle No.". When a value is set in the parameter "#43107 S_disp_name" (Spindle display name), "S + Spindle display name" is displayed according to its setting.
(5)	Load current (%)	This displays the NC axis load current.
(6)	Load(%)	This displays the spindle motor load. When the parameter "#3024 sout" (Spindle connection) is set to any of "2" to "5", the value which is set in the signal "Interface of load meter for analog I/F spindle" (R7012) is displayed.

Menus

Menus	Details
Close	This closes the pop-up window and quits this function.

Note

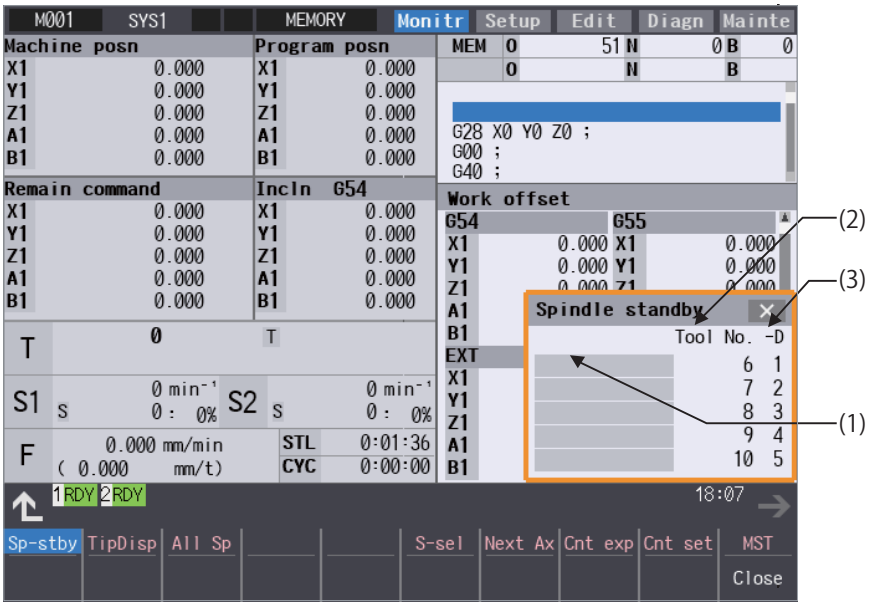
(1) Due to the specifications issued by MTB, pressing the menu [LdMeter] without setting ON load meter display causes an error.

10.2.11 Spindle, Standby Display

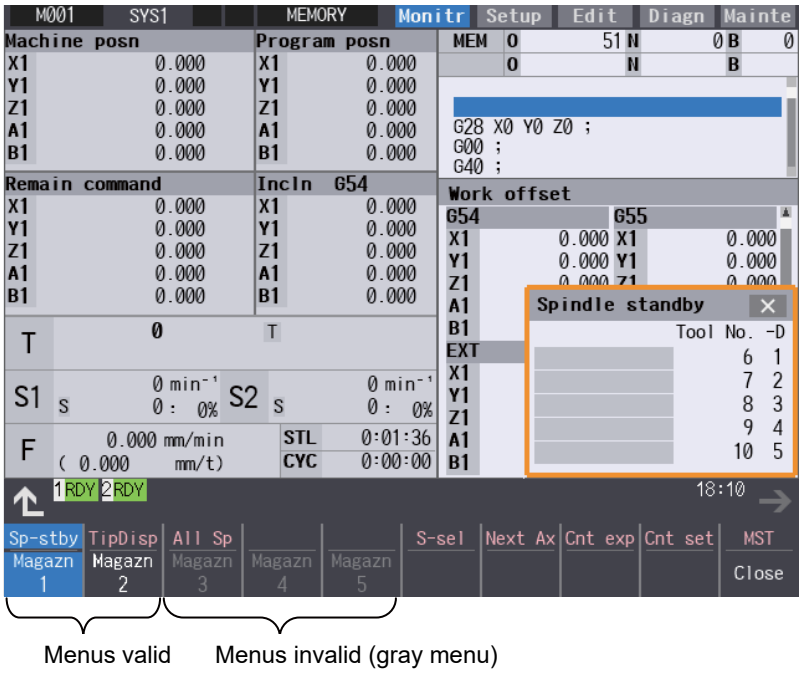
Monitr – Sp-stby (M system)

Monitr – Sp-stby (L system)

The current spindle tool No. and the standby tool No. (a maximum of 4 tools) are displayed.



<"#11054 Sp-stby disp type" is set to "1">



Display items

Display items		Details
(1)	Spindle standby	The name of this area differs depending on the output from the PLC program.
(2)	Tool No.	This displays spindle or standby tool No. Spindle/standby display for each magazine varies depending on the setting of the parameter "#11054 Sp-stby disp type".
(3)	-D	For D data function and purpose, refer to the instruction manual issued by the MTB.

Menus

Menus	Details
Magazn 1 : Magazn 5	This switches the spindle/standby display for each magazine. This menu can be switched when the parameter "#11054 Sp-stby disp type" is set to "1". When the parameter is set to "0", this menu is invalid (not displayed). The selectable menu differs according to the machine specifications. If there are two magazines, the menus [Magazn 1] and [Magazn 2] only are selectable. The other menus will be grayed out and non-selectable.
Close	This closes the pop-up window and quits this function.

Precautions

(1) The number of the standby tool to be displayed can be changed by changing the value of bit 8 to F of ATC control parameter (R10600).

(2) If the spindle/standby display is set not to be displayed by user PLC, when the menu [Sp-stby] is pressed, the operation message "Spindle/stndby tool display not possible" appears.

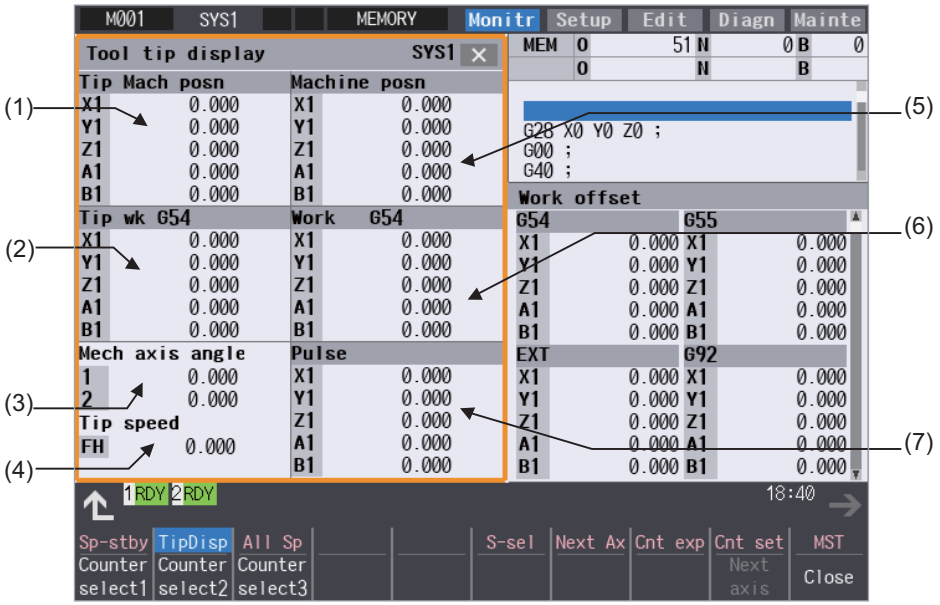
10.2.12 Tool Center Coordinate Display

Monitr – TipDisp

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

This function is an optional specification. Any of the following specifications is required.

- Tool center point control
- Tool length compensation along the tool axis
- Tool handle feed & interruption
- Workpiece installation error compensation
- Inclined surface machining command
- Simple inclined surface machining command
- 3-dimensional tool radius compensation (tool's vertical-direction compensation)
- 3-dimensional manual feed
- R-Navi



Display items

	Display items	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 wk G54	This displays the position of the tool center from the workpiece coordinate reference position in the selected workpiece coordinate system.
(3)	Mech axis angle	This displays the rotation angle when using the mechanical axis. 1st rotary axis angle: R2628 (L)/R2629 (H) 2nd rotary axis angle: R2630 (L)/R2631 (H)
(4)	Tip speed	This displays the movement speed of tool center. By the parameter "#1125 real_f", actual feedrate and command speed can be switched.
(5)	Counter display1	The counter type can be selected from the menu. The following counters are displayed by default. Counter display 1: Machine position counter Counter display 2: Workpiece position counter Counter display 3: Tool axis movement counter
(6)	Counter display2	
(7)	Counter display3	

Note

(1) Status abbreviations are displayed only in "Tip Mach posn" counter area. They are not displayed in other counter area.

(2) The data for 5 axes can be displayed in each counter. In case the number of valid axis is 6 or more, and you want to display the 6th and subsequent axes, press the menu [Next axis] to switch display.

Menus

Menus	Details
Counter select1 Counter select2 Counter select3	Select the counter type to display on "the counter display 1 to 3". It is valid after the power is turned ON again. The counters shown below can be selected. •Machine position counter •Workpiece coordinate position counter •Workpiece installation (*1) •Inclined surface (*2) •Tool axis movement
Next axis	This changes the displayed axes each counter.
Close	This closes the pop-up window and quits this function.

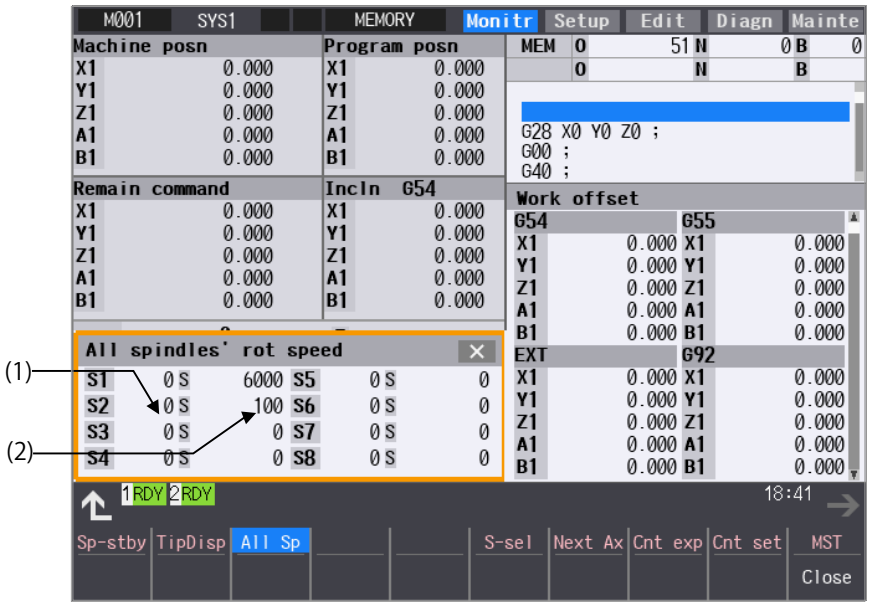
(*1) It cannot be selected when the workpiece installation error compensation specification is not valid.

(*2) This item cannot be selected when all of three optional specifications (inclined surface machining command, simple inclined surface machining command and R-Navi) are invalid.

10.2.13 All Spindles' Rotation Speed Display

Monitr – All Sp

The command rotation speed and actual rotation of all spindles are displayed.
This function only displays the value. It is not possible to operate here.



Display items

Display items		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

Menus	Details
Close	This closes the pop-up window and quits this function.

Precautions

(1) The number of spindles to be displayed depends on the setting of the parameter "#1039 spinno" (Number of spindles).

(2) When a single spindle is to be displayed, only "S" is displayed as the label without the number "1". However, when a value is set in the parameter "#43107 S_disp_name" (Spindle display name), the label display depends on its setting. (Example) When "A" is set in #43107, the label display becomes "SA".

10.3 Checking the Movement under Machining (Trace Screen)

Monitr – 

(M system)

Monitr – 

(L system)

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. Pressing [Trace ON] changes to the machine position trace mode and pressing [Tip traceON] changes to the tool center point trace mode.

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.

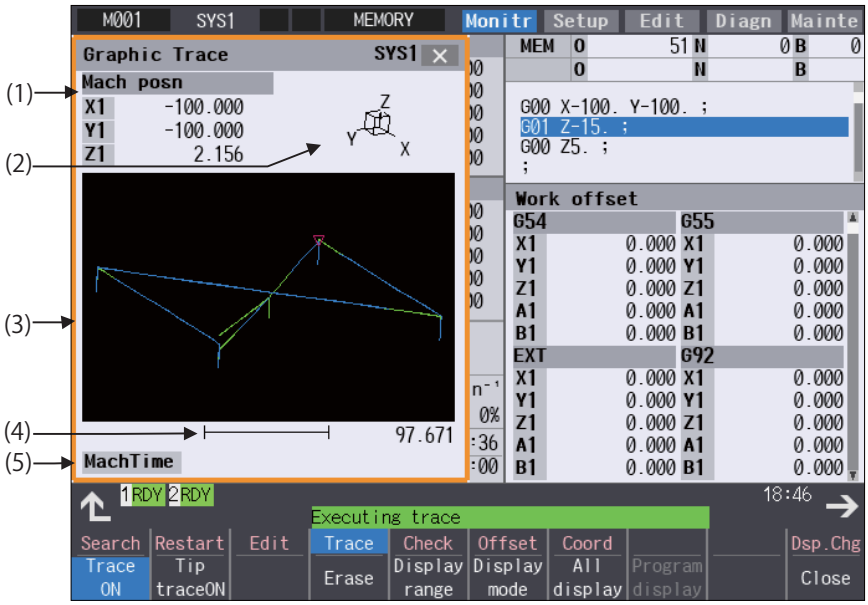
Using the menu [All display], normal display and whole display modes can be switched.

Note

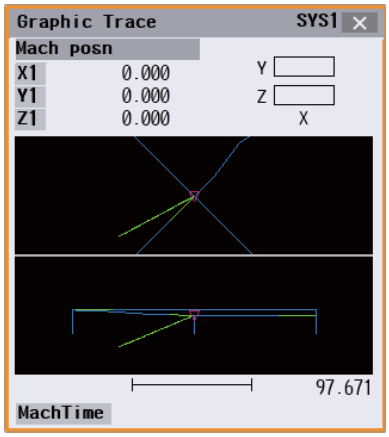
(1) The tool center point trace function is an optional specification. Any of the following specifications is required.

- Tool center point control
- Tool length compensation along the tool axis
- Tool handle feed & interruption
- Workpiece Installation Error Compensation
- Inclined surface machining command
- Simple inclined surface machining command
- 3-dimensional tool radius compensation (tool's vertical-direction compensation)
- 3-dimensional manual feed
- R-Navi

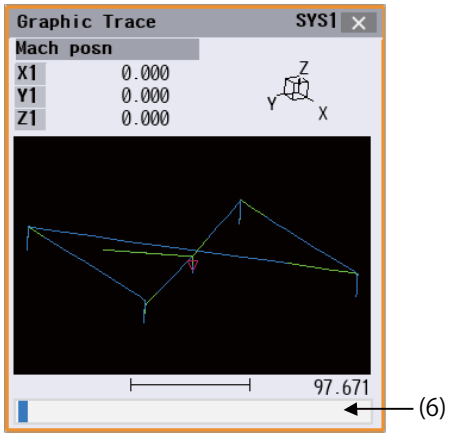
■Normal display ([All display] is OFF)



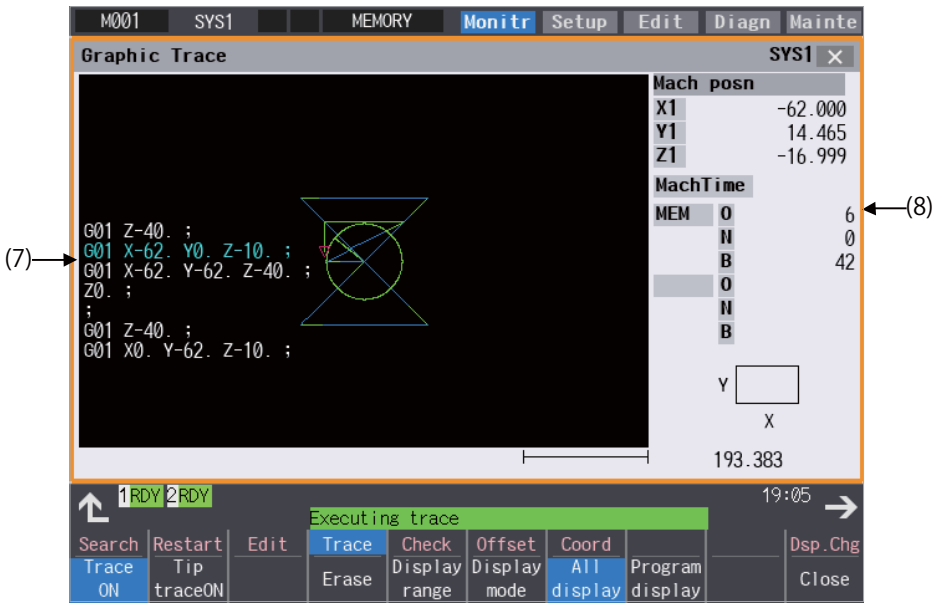
<2-plane drawing area>



<Displaying input area>



■Full-screen display ([All display] is ON)



Display items

Display items		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. A movement path is drawn as shown below: Trace mode #1231 set03/bit1 = 0: Machine position #1231 set03/bit1 = 1: Tool position Tool center point trace mode: Tool center machine position 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/bit4". Tool center point trace mode: The machine position of tool center point zero point
(4)	Scale	The display range scale is displayed. The unit is set to [mm/100pixcel] for metric system and [inch/100pixcel] for inch system.
(5)	Machining time display	The time required for machining is calculated and displayed. Nothing is displayed during tracing.
(6)	Input section	The input area appears when the menu [Display range] and [Display mode] are pressed. The scale value and display mode are set. The input area is hidden when the [INPUT] key is pressed.
(7)	Program display	The contents of the machining program that is in the process of drawing are displayed. The block currently being executed is displayed in blue characters. For 8.4-type/10.4-type display unit, the program display is not displayed on the normal display mode.
(8)	Program No. display	At the full screen display, the program number, sequence number, and block number which are currently being executed are displayed. While a subprogram is executed, its program number, sequence number, and block number are displayed.

Menus

Menus	Details
Trace ON	This activates the trace mode. If any program is currently running, the machine position path or tool position path is traced from the current position. If this menu is pressed during the trace mode, the trace mode will be turned OFF.
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 or tool 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. However, when the targeted specification (*1) is invalid, this menu is not displayed.
Erase	This erases the data in graphic drawing area displayed on the screen.
Display range	This changes the graphic drawing display range. When this menu is pressed, the menu changes to the display range change menu. The input area appears. It is also possible to designate the scale value with numerical input. When the display range is changed, the graphic data displayed on the screen is erased.
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 drawing mode: 1-plane, 2-plane, and 3-dimension. This changes the drawing plane and erases the graphic data displayed on the screen.
All display	This switches the normal display mode and the full-screen display mode.
Program display	This displays the currently executed program on the graphic drawing area. However, machining program contents being in the program lock state is not displayed. Only "%" is displayed. For 8.4-type/10.4-type display unit, this menu can be selected when the menu [All display] is active.
Rotate	This sets the viewpoint angle for the 3-dimension display mode. When the viewpoint angle is changed, the graphic data displayed on the screen is erased. However, this menu cannot be used for other than the 3-dimension display mode.
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.
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 menu [Trace ON] or [Tip traceON] is not highlighted, this is grayed out and non-selectable.
Next axis	This changes the axis to be displayed on the counter. This switches between "the basic axes IJK" and "C axis". <Note> ♦This menu is grayed out and non-selectable in the following cases: - When the control axis address other than C is specified in the parameter "#19405 Rotary ax drawing". - When the C axis is specified by the parameter "#1026 base_I", "#1027 base_J", or "#1028 base_K". - When the "Graphic check/trace rotary axis drawing" specification is invalid
Close	This closes the pop-up window and quits this function.

(*1) Targeted specifications

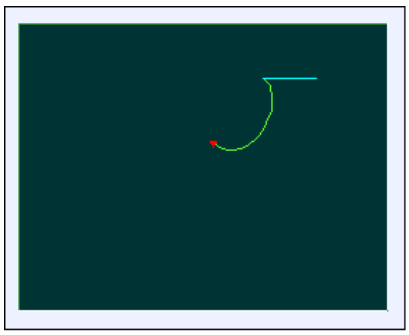
♦Tool center point control
♦Tool length compensation along the tool axis
♦Tool handle feed & interruption
♦Workpiece installation error compensation
♦Inclined surface machining command
♦Simple inclined surface machining command
♦3-dimensional tool radius compensation (tool's vertical-direction compensation)
♦3-dimensional manual feed
♦R-Navi

10.3.1 Displaying the Trace

Operation method

(1) Press the main menu [Trace ON].

The menu [Trace ON] is highlighted.
If [Tip TraceON] is highlighted, unhighlight it.
The machine position appears in the drawing area as a tool mark.
The counter selected by the menu [Counter type] 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 is drawn with a solid blue and green line.

Press the menu [Trace ON] again to cancel the trace.

Usually, paths are drawn at the machine position. Depending on the parameter setting, paths are drawn at the tool position.

#1231 set03/bit1 (Switch graphic coordinates)

Parameter value	Description
0	Paths are drawn at the machine position.
1	Paths are drawn at the tool position.

Note

(1) At the machine position, the length compensation and radius compensation are applied to the path.
At the tool position, only the length compensation is applied to the path. If the compensation direction is changed during the machining, such as the radius compensation, the drawn path does not match the machining path.

(2) Even if the setting of the parameter "#1231 set03/bit1" is changed, the type of the displayed counter is not switched.
Press the menu [Counter type] and select the counter to be displayed.

(3) This parameter does not support the tip position trace mode.

Color of the path drawn by tracing

Manual feed	Drawing at the tool position	Solid line (white)
	Drawing at the machine position	Solid line (blue)
Rapid traverse	Drawing at the tool position	Solid line (white)
	Drawing at the machine position	Solid line (blue)
Cutting feed	Drawing at the tool position	Solid line (white)
	Drawing at the machine position	Solid line (green)

10.3.2 Displaying the Machine Position of the Tool Center Point Trace

Operation method

(1) Press the main menu [Tip TraceON].

The menu [Tip TraceON] 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 the menu [Counter type] is displayed at the counter.
The message "Executing trace(tip posn)" appears.
After this, the tool center point path is drawn with graphics in the tool center point trace mode.

Press the menu [Tip traceON] again to cancel the machine position of the tool center point trace.

Note

(1) Even if the menu is unhighlighted, the displayed counter type does not change. However, when the trace window is closed and displayed again, the displayed counter type will be the trace mode counter.

Color of the path drawn by tracing

Manual feed	Solid line (blue)
Rapid traverse	Solid line (blue)
Cutting feed	Solid line (green)

10.3.3 Changing the Display Range

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

Operation method (Enlarging and reducing the drawing)

(1) Press the menu [Display range].

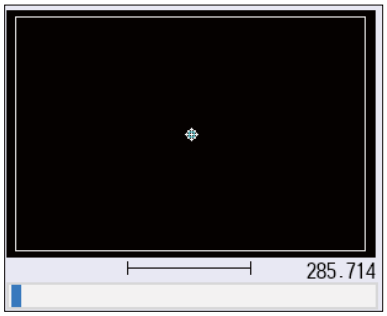
A white frame indicating the display range appears on the screen.
The display range input mode is activated.

(2) Press the menu [Zoom in] or [Zoom out].

The size of the white frame can be changed by using key operations.

To enlarge the figure:
Press the menu [Zoom in] or [-] key.
To reduce the figure:
Press the menu [Zoom out] 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 [INPUT] key.

The display scale changes.
By changing the display scale, the graphic data displayed on the screen is erased.

Note

(1) 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 menu [Display range]. → A white frame indicating the display range appears on the screen.
The display range input mode is activated.

(2) Press the menus [↑], [↓], [←], [→] 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 [INPUT] key. → The display position changes so that the cursor position (□) is the center of the area.
When display position is changed, the graphic data displayed on the screen is erased.
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.

Operation method (Centering)

(1) Press the menu [Display range]. → A white frame indicating the display range appears on the screen.
The display range input mode is activated.

(2) Press the menu [Centering]. → 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.

Operation method (setting the display scale)

(1) Select a scale value, and press the [INPUT] key.
(Example) 450 [INPUT]

Note

(1) The setting range of scale value is as shown below.

#1003 Input setup unit	#1014 Initial inch	
	0 (mm)	1 (inch)
B	0.100 to 9999.999	0.0100 to 9999.9999
C	0.0100 to 9999.9999	0.00100 to 9999.99999
D	0.00100 to 9999.99999	0.000100 to 9999.999999
E	0.000100 to 9999.999999	0.0000100 to 9999.9999999

(2) The set scale value is maintained even after the power is turned OFF and ON.

(3) If a scale value is entered while the screen is in the two-plane display mode, such as [XY/XZ], the display range (scale value) is changed for the two planes. The operation method is the same as the 1-plane display mode.

10.3.4 Changing the Display Mode

The graphics display mode includes the 1-plane, 2-plane and 3D mode. When the menu [Display mode] 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.

(2) These set display mode is maintained even after rebooting the machine.

(3) The display mode can be set independently for the trace function and program check function. These settings are not linked with one another.

(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

Menus	Description
XY	This changes to the 1-plane display mode configured of X-Y. The X axis is displayed in the horizontal axis, and the Y axis is displayed in the vertical axis.
XZ	This changes to the 1-plane display mode configured of X-Z. The X axis is displayed in the horizontal axis, and the Z axis is displayed in the vertical axis.
YX	This changes to the 1-plane display mode configured of Y-X. The Y axis is displayed in the horizontal axis, and the X axis is displayed in the vertical axis.
YZ	This changes to the 1-plane display mode configured of Y-Z. The Y axis is displayed in the horizontal axis, and the Z axis is displayed in the vertical axis.
ZX	This changes to the 1-plane display mode configured of Z-X. The Z axis is displayed in the horizontal axis, and the X axis is displayed in the vertical axis.
ZY	This changes to the 1-plane display mode configured of Z-Y. The Z axis is displayed in the horizontal axis, and the Y axis is displayed in the vertical axis.
XY/XZ	This changes to 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.
YX/YZ	This changes to 2-plane display mode configured of Y-X and Y-Z. The Y axis is displayed in the horizontal axis, and the X and Z axes are displayed in the vertical axis.
XYZ	This changes to the 3D display mode. A cube is displayed on the lower right of the screen.

Operation method (Changing the display mode)

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

(2) Select the mode to display and press the menu.
(Example) [XY/XZ]

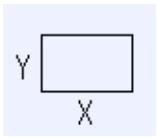
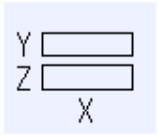
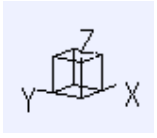
Operation method (Changing the display axis name)

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

(2) Set the axis name and press the [INPUT] key.
XYC [INPUT] → XYC appears as the axis name in the display mode. The X axis, Y axis and C axis are the three axes for drawing.

There are three types of display mode: 1- plane, 2- plane, and 3D. The display mode currently selected is shown with an image. The selected axis name displays for the image axis name.

Display example

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

10.3.5 Changing the Display Angle

This is used to set the graphic drawing display angle when in 3-dimension display mode.

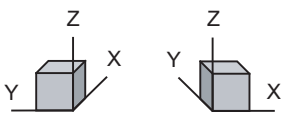
Note

(1) When the display angle is changed, the graphics displayed up to that point are erased.
(2) The set display angle is maintained even after rebooting the machine.
(3) The display angle can be set independently at the trace function and program check function. These settings are not linked with one another.
(4) The set display angle is maintained even after rebooting the machine.

Operation method

(1) Press the menu [Rotate].

(2) Use the menus [↑], [↓], [←], [→] to adjust the display angle.
The cursor movement keys ([↑], [↓], [←], [→]) can also be used to rotate. → Rotate the cubic display in the lower right of the screen using the operation keys.



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

10.3.6 Displaying the Machining Program on Checking

When pressing the menu [Program display], the contents of the machining program on checking is displayed on the drawing area.
When pressing the menu [Program display] again, the displayed program is erased.
Note that this menu can be used only when the full-screen display mode is selected.

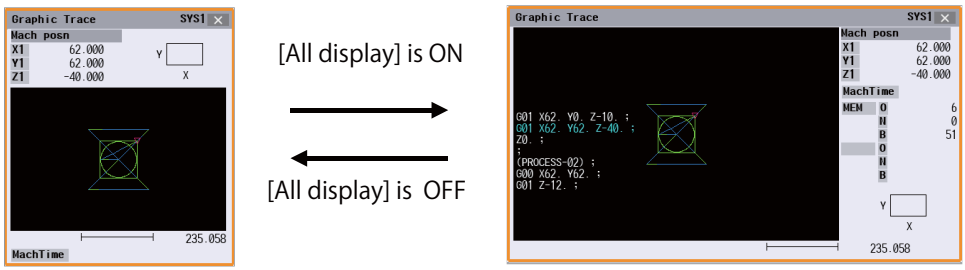
Note

(1) The selected display mode is maintained even after rebooting the machine.
(2) The program display mode is applied to the trace function and program check (2D) function commonly.
(3) For 8.4-type/10.4-type display unit, the menu [Program display] can be selected when the menu [All display] is active.
(4) The program display mode is applied commonly to all part systems.

10.3.7 Switching the Full-screen Display Mode

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

Press this menu again to return the normal display.



Note

(1) The selected display mode is maintained even after rebooting the machine.
(2) The full-screen display mode is applied to the trace function and program check (2D/3D solid) function commonly.
(3) The full-screen display mode is applied commonly to all part systems.

10.3.8 Switching the Axis Movement Direction (+/-)

Graphic trace draws the path according to the axis movement direction designated by the parameter "#1495 grf_ax_direction". The movement direction can be set for each axis. For details, refer to "4.1.8 Switching the Axis Movement Direction (+/-)".

10.3.9 Drawing a Path Corresponding to the Rotary Axis

If a rotation axis (C axis) is designated in the parameter "#19405 Rotary ax drawing", the graphic trace draws a path corresponding to the rotary axis. The rotary axis can be set for each part system. For details, refer to "4.1.13 Drawing a Path Corresponding to the Rotary Axis (L System)".

10.3.10 Precautions for Tracing

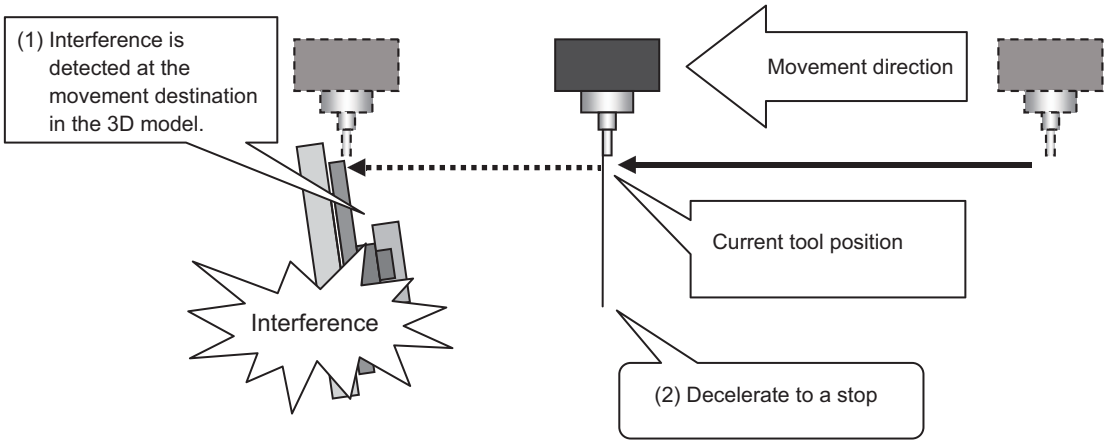
(1) When the command speed increases in the trace mode, the shape being drawn with graphics will become distorted. Especially, correct graphic drawing is not possible during machine lock.
(2) If program check (2D) is executed during the trace mode, the trace mode will be canceled.
(3) If trace is executed during program check mode, the program check will be canceled.

For other precautions, refer to "4.1.18 Notes for Graphic Check (2D)".

10.4 3D Machine Interference Check (M System) [M800VW/M80VW Series]

Monitr – 3D MON

This function checks the machine operation in real time by using 3D model of machine, tool and jig. The movement destination of the machine is checked in advance by using the 3D model, an alarm is displayed as soon as an interference is detected, and all the axes are decelerated to stop.

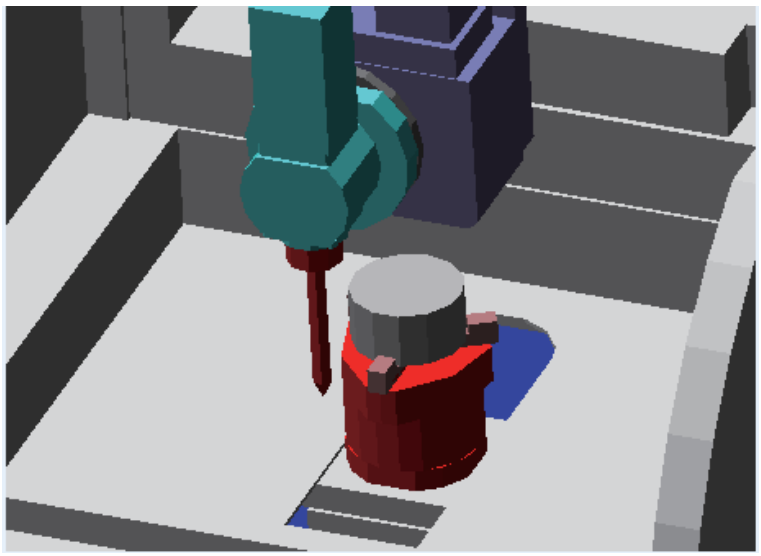


The 3D model of machine and tool is displayed on the monitor screen in real time so that the actual positional relation of the machine can be figured out.

The interference check is performed between the following items, and the interference part in the 3D model is shown in yellow or red.

- Machine and machine
- Machine and tool
- Machine and jig
- Tool and jig

The interference with workpiece is not checked. This function can be used in single part system of M system.



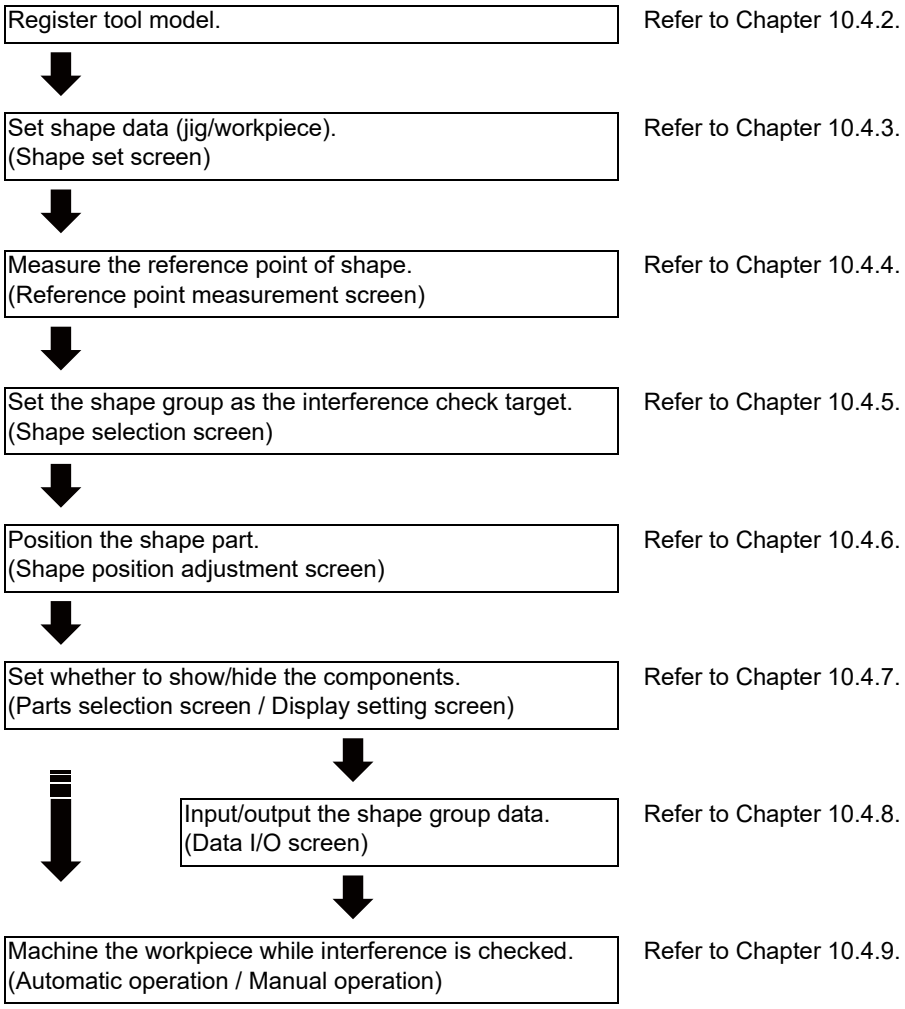
Preparation by MTB

To use this function, MTB needs to prepare 3D model of machine, tool, jig, workpiece, etc. and set some parameters.

The software sold separately is required to create the 3D model.

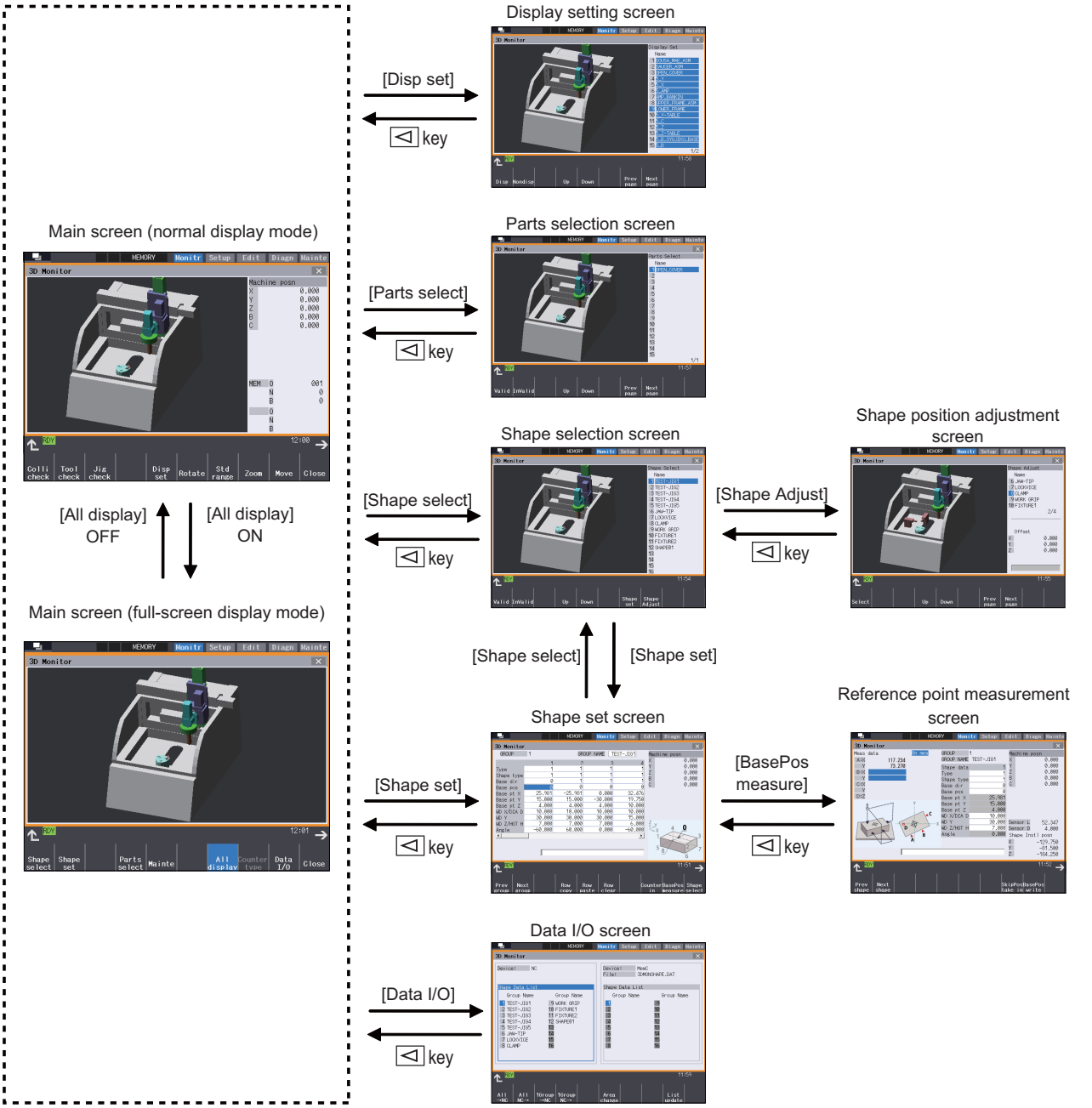
Flow from setup before machining to operation

MTB registers/sets machine model.



Screen configuration

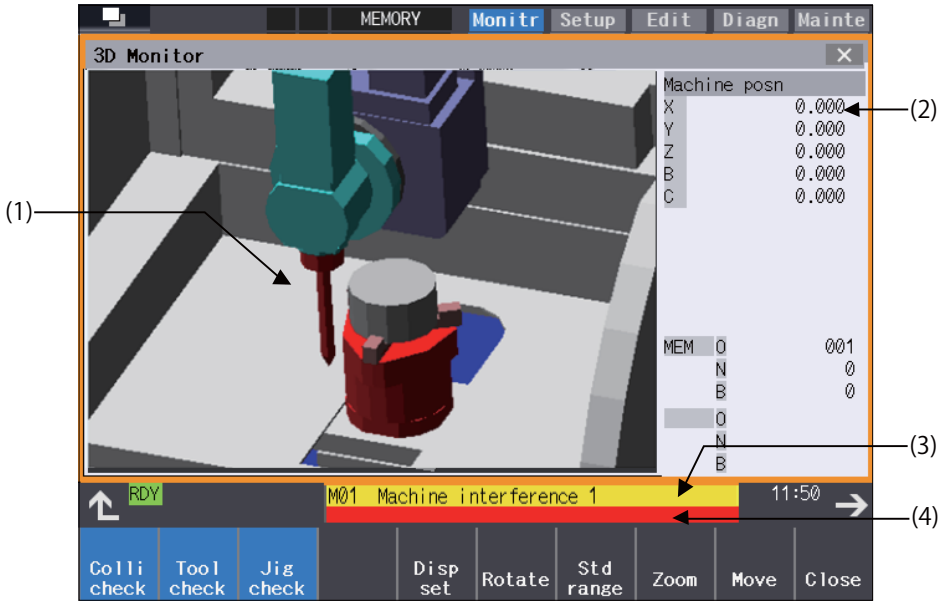
The screen configuration of 3D machine interference check function is as follows.



10.4.1 3D Monitor Screen (Main Screen)

The 3D model (machine, tool, jig, and workpiece) is displayed in real time according to the indication of the machine position.

When the interference occurs, the interference part in the 3D model will be shown in yellow or red.



Display items

Display items		Details
(1)	3D model display area	3D model is displayed.
(2)	Counter	The counter currently selected is displayed. Machine position, current position and set base position can be selected by the [Counter type] menu.
(3)	Alarm	The highest priority message among the currently occurring alarms or warnings is displayed.
(4)	Operation message	The operation message is displayed. The name of interference part is displayed during the interference.

Menus

Menus	Details
Colli check	This enables or disables the machine interference check. The machine interference check is enabled when the power is turned ON.
Tool check	This enables or disables the tool check. The tool check is enabled when the power is turned ON.
Jig check	This enables or disables the jig check. The jig check is enabled when the power is turned ON.
Disp set	This displays the display setting screen. Refer to "10.4.7.2 Display Setting Screen" for details of the display setting screen.
Rotate	This activates the 3D model rotation mode. The sub menus [Up]/[Down]/[Left]/[Right] change the viewpoint of the model.
Std range	This initializes the viewpoint angle and the scale of the model display area.
Zoom	This activates the 3D model zoom mode. The sub menus [Zoom In]/[Zoom Out] change the size of the model. The same operation can be performed with the [-]/[+] keys and by pinching in / pinching out on the screen. (*1)

Menus	Details
Move	This activates the mode to move the 3D model position. The sub menus [Up]/[Down]/[Left]/[Right] move the model.
Close	This returns the screen from the 3D monitor screen to the monitor screen.
Shape select	This displays the shape selection screen. A shape group (jig and workpiece) for the interference check can be selected. Refer to "10.4.5 Setting the Shape Group as the Interference Check Target" for details of the shape selection screen.
Shape set	This displays the shape set screen. The shapes of workpiece and jig can be defined. Refer to "10.4.3 Setting the Shape Data (Jig/Workpiece)" for details of the shape set screen.
Parts select	This displays the parts selection screen. Parts as the interference check target can be selected from the optional parts registered by MTB. Refer to "10.4.7.1 Parts Selection Screen" for details of the parts selection screen.
Mainte	This displays the maintenance menus. This menu is used by MTB.
All display	This hides the machine position and currently executing program, and enlarges the 3D model display area. The [Counter type] menu cannot be selected during full-screen display mode.
Counter type	This displays the sub menus of counter type ([Machine posn]/[Curnt posn]/[SetBase posn]). This cannot be selected during full-screen display mode.
Data I/O	This displays the data I/O screen. The shape data of jig/workpiece can be input/output. Refer to "10.4.8 Inputting/outputting the Shape Group Data" for details of the data I/O screen.

(*1) The method to enlarge/reduce can be switched by the parameter "#1251 set23/bit7" (3D machine interference check: pinch gesture).
0: Pinch in to enlarge the 3D model, and pinch out to reduce the model.
1: Pinch in to reduce the 3D model, and pinch out to enlarge the model.

10.4.2 Registering a Tool Model

Tool model is defined by registering the tool data.

To set the tool as the interference check target, the tool compensation amount and tool data need to be registered on the tool compensation screen and tool management screen in advance. The data referred to to define the tool diameter and length of tool model is as follows depending on the setting value of the parameter "#19723 3D intrf tooltype".

	#19723 = 0	#19723 = 1
Diameter	Tool data set on the tool management screen	Tool compensation amount set on the tool compensation screen (*1)
Length		Tool compensation amount set on the tool compensation screen
Other shape data		Tool data set on the tool management screen

(*1) For the tool compensation type I, tool data set on the tool management screen is referred to.

The registration data necessary for 3D interference check are tool No., tool type, each dimension in the tool shape (A to H in the tool shape drawing).

For details, refer to "6.4 Setting Tool Compensation Amount (Tool Offset Screen)", "6.5 Setting the Tool Management Data (Tool Management Screen)".

10.4.3 Setting the Shape Data (Jig/Workpiece)

Jig models and workpiece models are set by entering the shape data on the shape set screen.

The jig check (interference check between jig and machine and between jig and tool) can be performed by entering the shape data of the jig model.

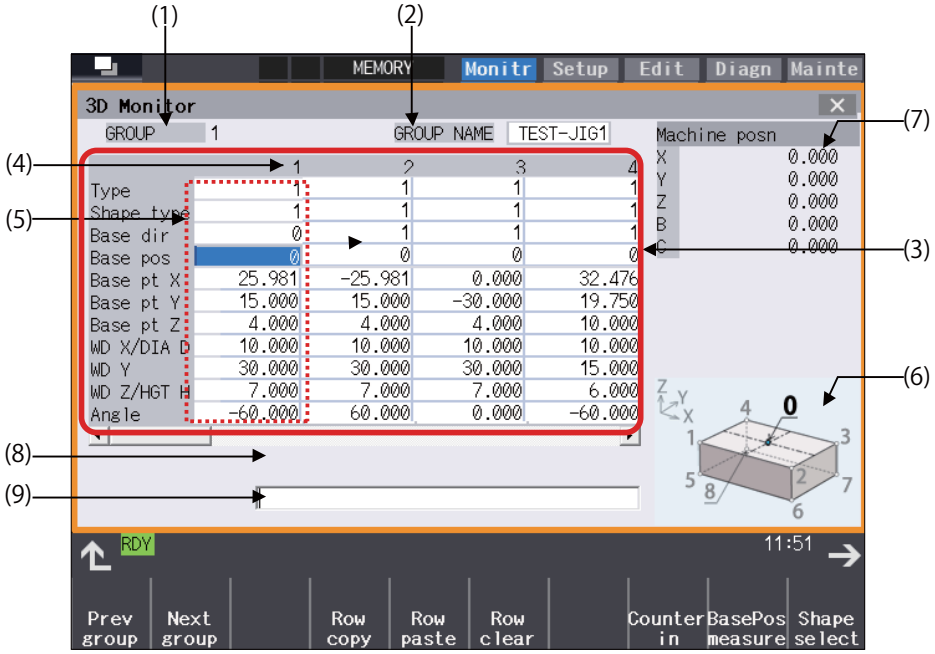
The workpiece image can be displayed on the 3D monitor screen by entering the shape data of the workpiece model.

Press the [Shape set] menu on the main screen to display the shape set screen.

Register a combination of jig and workpiece to be used together as a group.

Register the jig and workpiece shapes by combining blocks and pillars.

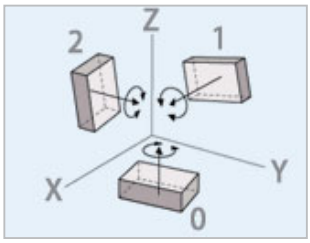
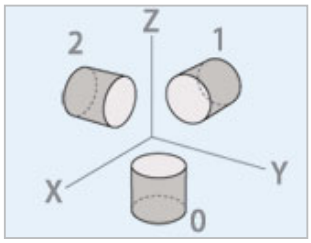
To set the shape data, select a group No. for each shape group, enter the group name, and enter the type, width and layout for each shape that configures the jig and workpiece.



Display items

	Display items	Details
(1)	GROUP	The No. of the selected shape group is displayed. There are the shape groups 1 to 16, that is, 16 groups in total.
(2)	GROUP NAME	The name of the selected shape group is displayed. Move the cursor to the "GROUP NAME" field to change the name (10 or less alphanumeric characters). To erase it, enter a space.
(3)	Shape setting list	The shape data in the shape group is set/displayed. Up to 20 sets of shape data can be registered for one group.
(4)	Shape No.	The shape data No. is displayed.
(5)	Shape data	Set the shape data for each part that configures the jig or workpiece. Refer to [Data setting range of the shape setting list] described later for the setting contents.
(6)	Guide drawing	The guide drawing is displayed according to the shape type depending on the cursor position where the data is being entered.
(7)	Counter	The counter of machine position, current position or set base position selected by the [Counter type] menu is displayed.
(8)	Help display	The guidance according to the input data is displayed depending on the cursor position where the data is being entered.
(9)	Input area	The input data is displayed.

[Data setting range of the shape setting list]

Setting item	Details	Setting range
Type	Select jig or workpiece.	1: Jig 2: Workpiece
Shape type	Specify the shape type. When the shape type is changed, the measurement data and shape data are erased. [Block]  [Pillar]  	1: Block 2: Pillar
Base dir	Select a plane where the shape is arranged.	0: Shape on XY plane 1: Shape on YZ plane 2: Shape on ZX plane
Base pos (*1)	Set the reference point when the shape type is block. Check the reference point position (0 to 8) on the guide drawing.	0 to 8
Base pt X Base pt Y Base pt Z	Set the coordinate value of "Base pos" on the set base coordinate system. For details, refer to "10.4.3.1 Arranging the Shape Data". It is also possible to set the measurement value by using a touch probe. For details, refer to "10.4.4 Measuring a Reference Point of Shape".	-999999.999 to 999999.999 (mm)
WD X/DIA D	Set the width as follows when the shape type is block. X-direction width when the XY plane is selected Y-direction width when the YZ plane is selected Z-direction width when the ZX plane is selected Set the shape diameter when the shape type is pillar.	0.001 to 999999.999 (mm)
WD Y (*2)	Set the block width as follows. Y-direction width when the XY plane is selected Z-direction width when the YZ plane is selected X-direction width when the ZX plane is selected	
WD Z/HGT H	Set the shape height.	
Angle (*2)	Set the inclination when the shape is block. For details, refer to "10.4.3.2 Entering an Inclination Angle (Block)".	-89.999 to 89.999 (°)

(*1) It cannot be set when the shape type is pillar. The reference point position is the center of the top face.

(*2) It cannot be set when the shape type is pillar.

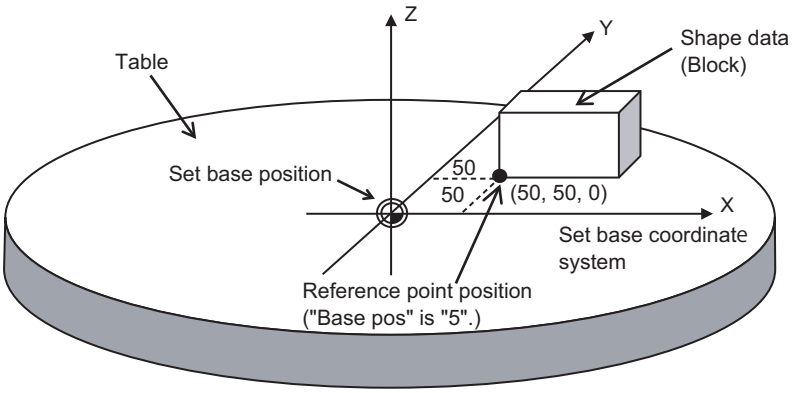
Menus

Menus	Details
Prev group Next group	These switch the displayed shape group.
Row copy	This copies the data at the cursor position. The copied data cell is displayed in light blue. The copied data is effective until the screen is switched.
Row paste	This pastes the data of the copied column to the cursor position.
Row clear	Press this menu, move the cursor to the column of the shape data to be deleted, and press the [INPUT] key. A deletion confirmation message appears. Press the [Y] key to clear all the data in the column.
Counter in	When the cursor is in "Base pt X", "Base pt Y" or "Base pt Z", the counter value of the same axis is imported to the input area.
BasePos measure	This displays the reference point measurement screen for the shape data where the cursor is currently located. Setting with a touch probe, with skip position, or by manual input is possible on the reference point measurement screen. Refer to "10.4.4 Measuring a Reference Point of Shape" for details of the reference point measurement screen.
Shape select	This displays the shape selection screen. A shape group (jig and workpiece) for the interference check can be selected. Refer to "10.4.5 Setting the Shape Group as the Interference Check Target" for details of the shape selection screen.
Up Down Left Right	These move the cursor. When the [Up] menu is pressed while "Type" is selected, the cursor moves to "GROUP NAME". When the [Down] menu is pressed while "GROUP NAME" is selected, the cursor moves to "Type" in the leftmost column.
Counter type	This displays the sub menus of counter type ([Machine posn]/[Curnt posn]/[SetBase posn]). The set base position counter indicates the position on the set base coordinate system.

10.4.3.1 Arranging the Shape Data

While all the machine axes are at zero point, set the coordinate value of "Base pos" (0 to 8) of the shape setting list, in "Base pt X", "Base pt Y" and "Base pt Z". The zero point of the set base coordinate system is as specified by MTB.

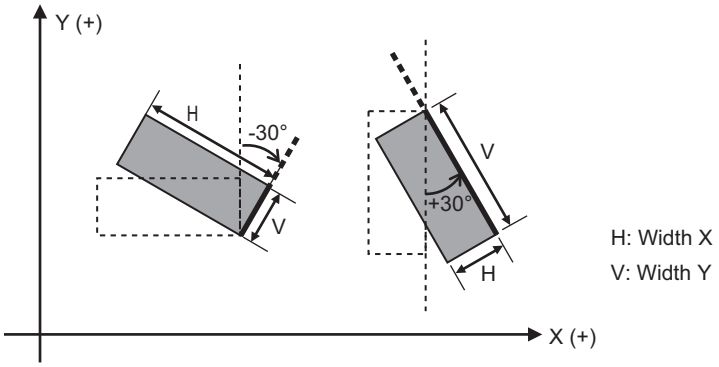
(Example) When "5" is specified in "Base pos" of the block shape data and the position of "Base pt X", "Base pt Y" and "Base pt Z" is set as (50, 50, 0), the shape data is arranged as follows.



10.4.3.2 Entering an Inclination Angle (Block)

When the shape type is block and the shape is not parallel to the coordinate axis, enter the inclination angle in the "Angle" field of the shape setting list. The inclination angle is the one between the axis perpendicular to the setting face and the side in the depth (WD Y) direction. The counterclockwise direction is positive.

(Example) When the setting face is the XY plane



"H: Width X" and "V: Width Y" are the same on the YZ plane and the ZX plane.

10.4.4 Measuring a Reference Point of Shape

"Base pt X", "Base pt Y", "Base pt Z" and "Angle" of the shape data can be not only directly entered but also set by reference point measurement using the touch probe.

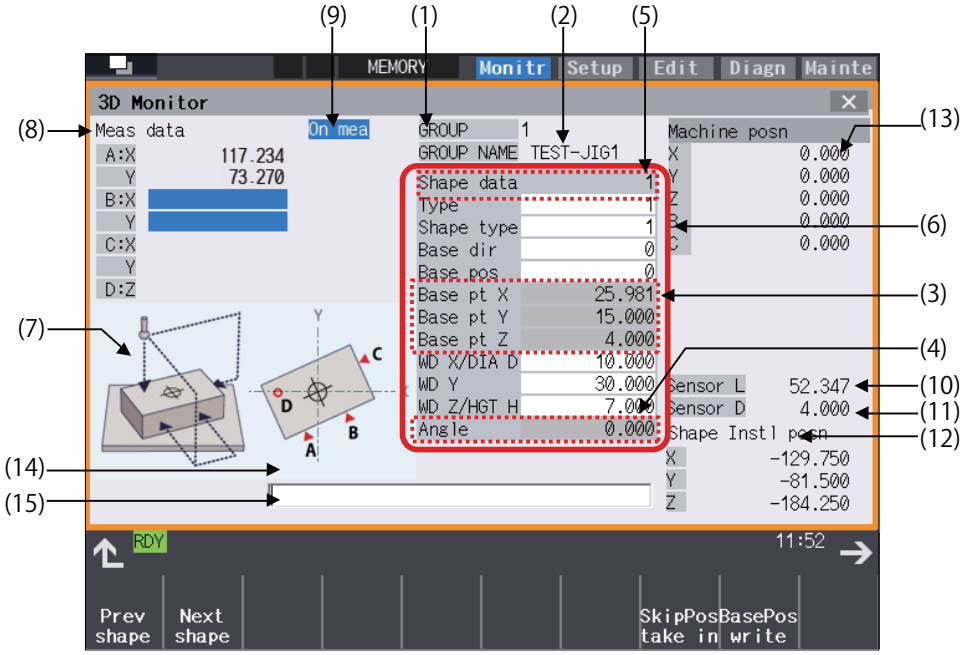
On the reference point measurement screen, after entering the shape data, the position is measured by moving the touch probe mounted to the spindle to make a contact with the jig/workpiece using manual feed or handle feed.

The measurement is performed while all the rotary axis machine positions are 0 degrees.

However, for the following rotary axes of 5-axis machine, the measurement is possible even when the position in the machine coordinate system is not 0 degrees.

- Tool-side 1st rotary axis (tool base-side rotary axis) for tool tilt type
- Table-side 2nd rotary axis (table workpiece-side rotary axis) for table tilt type
- Table workpiece-side rotary axis for combined type

Press the [Shape set] - [BasePos measure] menu on the main screen to display the reference point measurement screen.



Display items

	Display items	Details
(1)	GROUP	The No. of the shape group to be set is displayed.
(2)	GROUP NAME	The name of the shape group to be set is displayed.
(3)	Base pt X Base pt Y Base pt Z	The reference point of the shape which is currently being set is displayed. Press the [BasePos write] menu to set it. The reference point data is calculated from the shape data and measurement data. The measurement axes are those set in the parameters "#1026 base_I", "#1027 base_J" and "#1028 base_K".
(4)	Angle	The inclination angle from the base axis is displayed. Press the [BasePos write] menu to set it. The inclination angle is calculated from the shape data and measurement data.
(5)	Shape No.	The No. of the shape data which is currently being set is displayed.
(6)	Shape data	The shape data which is currently being set is displayed. Set each data before measurement. Refer to "10.4.3 Setting the Shape Data (Jig/Workpiece)" for setting contents.
(7)	Guide drawing	The guide drawing is displayed according to the shape type depending on the cursor position where the data is being entered.

Display items	Details
(8) Meas data	Sensor input and skip position import are enabled for the point where the cursor is displayed. The coordinate value on the set base coordinate system obtained by [BasePos write] or [Counter in] is displayed. The counter axis name varies depending on the setting of the "Base dir". When the "Base dir" is not set, the axis name is not displayed.
(9) "On mea" display	"On mea" is displayed when the measurement switch is ON.
(10) Sensor L	The length to the tip of the touch probe is displayed. This is set by the parameter "#8701 Tool Length".
(11) Sensor D	The diameter of the touch probe is displayed. This is set by the parameter "#8702 Tool Dia".
(12) Shape Instl posn	The set base position specified by MTB is displayed with the value of machine coordinate system.
(13) Counter	The counter of machine position, current position or set base position selected by the [Counter type] menu is displayed.
(14) Help display	The guidance according to the input data is displayed depending on the cursor position where the data is being entered.
(15) Input area	The input data is displayed.

Menus

Menus	Details
Prev shape Next shape	These switch the group No. of the shape group to be displayed.
SkipPos take in	This reads the current machine position as the skip position when the simple measurement without using the touch probe is performed. When the skip position is read, the set base position is displayed at the current cursor position. This menu is enabled when the cursor is moved to the measurement data.
BasePos write	This sets the reference point data and inclination angle using the measurement data and shape data. When there is unset data in the measurement data and shape data, an error occurs.
Up Down Left Right	These move the cursor position.
Counter type	This displays the sub menus of counter type ([Machine posn]/[Curnt posn]/[SetBase posn]).

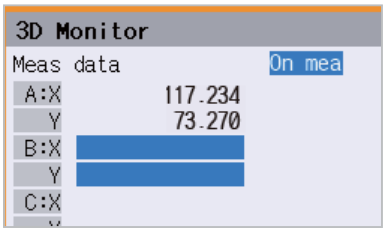
Operation method

To use a touch probe for measurement, the machining parameters (for each part system) #8701 to #8708 need to be set in advance.

(1) When there is unset item or item to be changed in the shape data, set it before measurement.

(2) Turn ON the measurement switch. → "On mea" is displayed at the top of the screen.

(3) Move the cursor to the measurement data display area by the [Left] menu, and then bring the touch probe into contact with the point A in the guide drawing of the jig/workpiece.
When the touch probe is not used, bring the tool into contact with the jig/workpiece, and then press the [SkipPos take in] menu. → The contacted position is imported into the point A field, and the cursor moves down to the point B field.



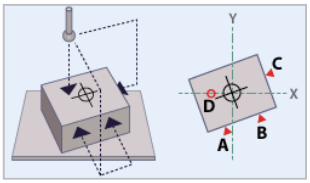
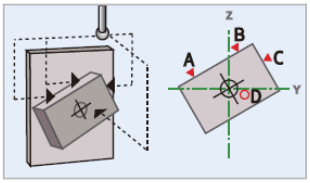
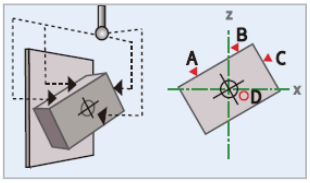
(4) Measure the points B to D in the same way.
The measurement point can be selected by pressing the [Up]/[Down] menus to move the cursor.

(5) Press the [BasePos write] menu after importing the points A to D. → "Base pt X", "Base pt Y", "Base pt Z" and "Angle" are set based on the measurement data and shape data.
The set data is reflected to the 3D model display area.
Check the shape data layout.
When there is unset value, the message "Can't write setting value" appears.

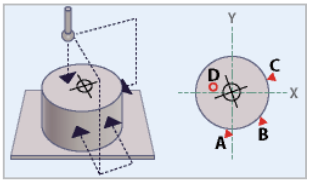
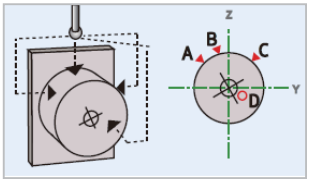
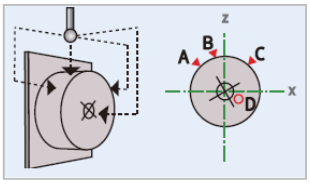
(6) Turn OFF the measurement switch.
Before automatic operation, check whether the correct position is set.

For the reference point measurement, the contact point differs depending on the shape (block/pillar).

[Contact point of block]

Plane	Contact point	Outline
XY plane		<Side> Contact the jig/workpiece at two points (A, B) on one of two adjacent sides and one point (C) on the other side. For YZ plane and ZX plane, the two points (A, B) and one point (C) must be on the upper two faces of the jig/workpiece. <Top> Contact the jig/workpiece at one arbitrary point (D). <When the touch probe is used on YZ plane or ZX plane> When the reference point is measured at the A, B, and C positions, contact the touch probe from the following axis direction. YZ plane: Y direction ZX plane: X direction When the touch probe is contacted from Z direction, measurement cannot be performed correctly.
YZ plane		
ZX plane		

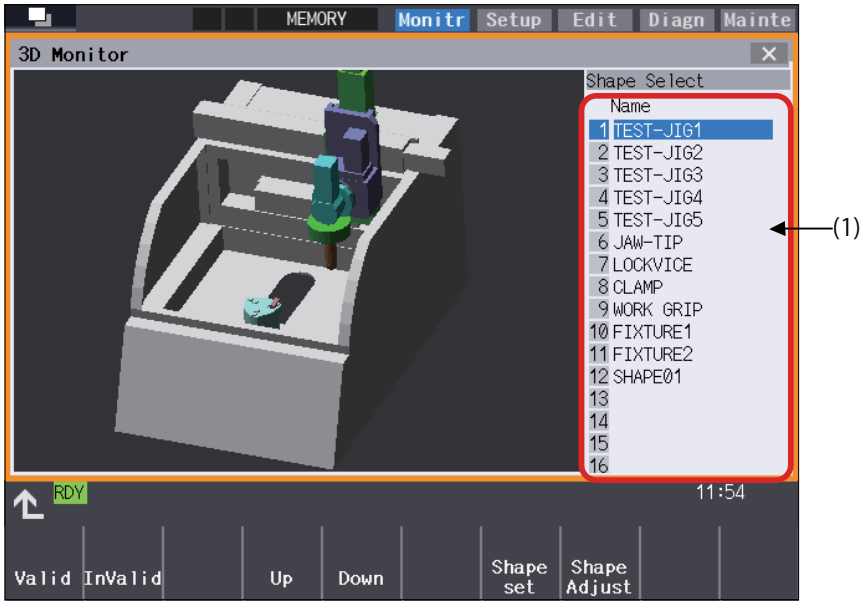
[Contact point of pillar]

Plane	Contact point	Outline
XY plane		<Side> For XY plane, contact the jig/workpiece at three arbitrary points. For YZ plane and ZX plane, contact the jig/workpiece at two points (A, B) on one of two adjacent quadrants and one point (C) on the other quadrant. <Top> Contact the jig/workpiece at one arbitrary point.
YZ plane		
ZX plane		

10.4.5 Setting the Shape Group as the Interference Check Target

Set the shape group (jig/workpiece) as the interference check target on the shape selection screen.

Press the [Shape select] menu on the main screen to display the shape selection screen.



Display items

Display items	Details
(1) List of shape group	The names of the shape groups that can be switched between target and non-target of interference check are displayed. Select a shape group by the [Up]/[Down] menus, and press the [Valid] menu to set the interference check target. Only one group can be selected. The name of the selected shape group is highlighted. The set condition remains after the power is turned ON again.

Note

(1) There may be a case where the interference check target cannot be switched on this screen depending on MTB settings.

Menus

Menus	Details
Valid	This sets the jig data set to the shape group at the cursor position, as the jig check target. <Note> •This cannot be operated during automatic operation.
InValid	This excludes the jig data set to the shape group at the cursor position, from the jig check target. <Note> •This cannot be operated during automatic operation.
Up Down	These move the cursor position.
Shape set	This displays the shape set screen. Refer to "10.4.3 Setting the Shape Data (Jig/Workpiece)" for details of the shape set screen.
Shape Adjust	This displays the shape position adjustment screen. Refer to "10.4.6 Positioning the Shape Part" for details of the shape position adjustment screen.

10.4.6 Positioning the Shape Part

When the jig setting position is changed due to replacement of pallet or jig, the machine model is compensated to the actual jig position and interference check is performed.

Set the change amount of the jig setting position in "Offset" on the shape position adjustment screen, and adjust the position for each shape group. Movement is possible in X, Y and Z directions of assembly coordinate system of the machine model.

Note

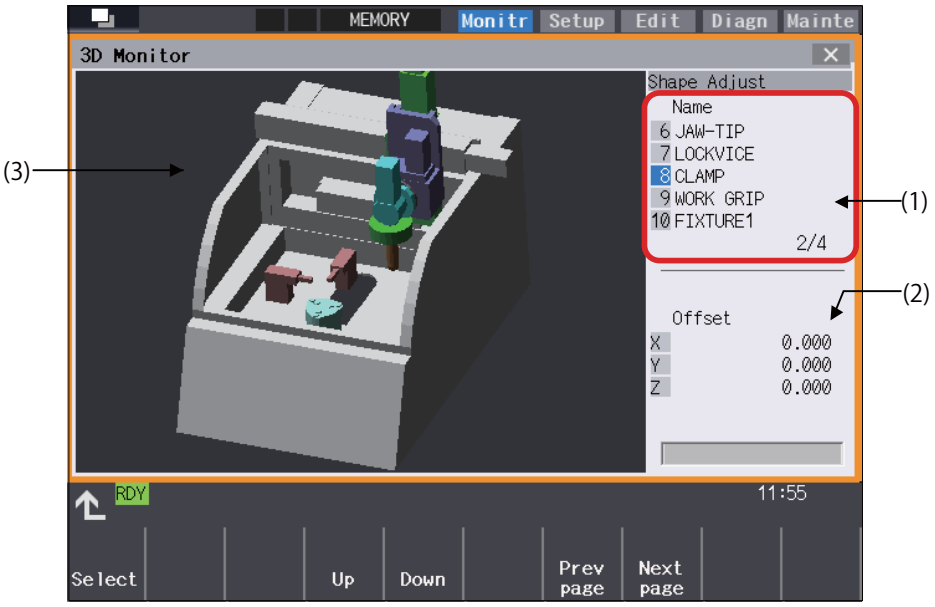
(1) Movement is not possible in A, B or C axis direction.

(2) Only some of the components in the shape group cannot be moved.

(3) The size cannot be changed.

(4) The set offset amount is reflected to the actual interference check. When the set value is incorrect, the interference check is not performed correctly. Check whether the set value is appropriate to perform the operation.

Press the [Shape select] - [Shape Adjust] menu on the main screen to display the shape position adjustment screen.



Display items

Display items			Details
(1)	List of shape group		The names of the shape groups that can be adjusted are displayed. Select a shape group by the [Up]/[Down] menus, and press the [Select] menu to select the group. Only one group can be selected. The name of the selected shape group is highlighted.
(2)	Offset	X Y Z	Set the difference between the jig/workpiece position and the jig/workpiece shape model. The axis name displayed here indicates the axis of assembly coordinate system of the machine model. Set the deviation between the jig/workpiece position and the jig/workpiece shape model for each axis at the relative position based on the shape model.
(3)	Machine model display		The machine model is displayed.

[Data setting range]

Setting item	Details	Setting range
Offset	Set the difference with the shape position data for each axis. <Note> ♦When there is a difference of 100 mm or more between the actual jig/ workpiece and the shape model position, the shape position data needs to be set again.	-99.999 to 99.999 (mm)

Menus

The menus displayed before the shape group is selected are the same as those on the shape selection screen. For details, refer to "10.4.5 Setting the Shape Group as the Interference Check Target".

Menus	Details
Select	This selects the shape group at the cursor position and changes the mode to the offset amount input mode. The selected shape group is highlighted, and the offset amount setting area becomes active. When this menu is pressed, the axis X of "Offset" is selected.
Up Down	These move the cursor position.
Prev page Next page	These switch the display page when the shape group cannot be displayed within one page.
Set	This fixes the set offset amount and returns the state to the part selected state. The offset amount set when the interference check is performed is reflected.
Cancel	This cancels the set offset amount and returns the state to the shape group selected state. The offset amount returns to the one before the [Select] menu is pressed.

Operation method

(1) Select a shape group by the [Up]/[Down] menus, and press the [Select] menu. The setting field of "Offset" becomes active.

(2) Select an axis by the [Up]/[Down] menus, and enter the difference between the actual jig/workpiece and the shape model position.

(*1) X/Y/Z indicate the axes of assembly coordinate system of the machine model.

<Note>

♦When the offset amount is 100 mm or more, set the shape position data again.

(3) Press the [Set] menu to reflect the set value to the shape model position.

To stop the setting of offset amount, press the [Cancel] menu.

10.4.7 Setting Whether to Show/Hide the Components

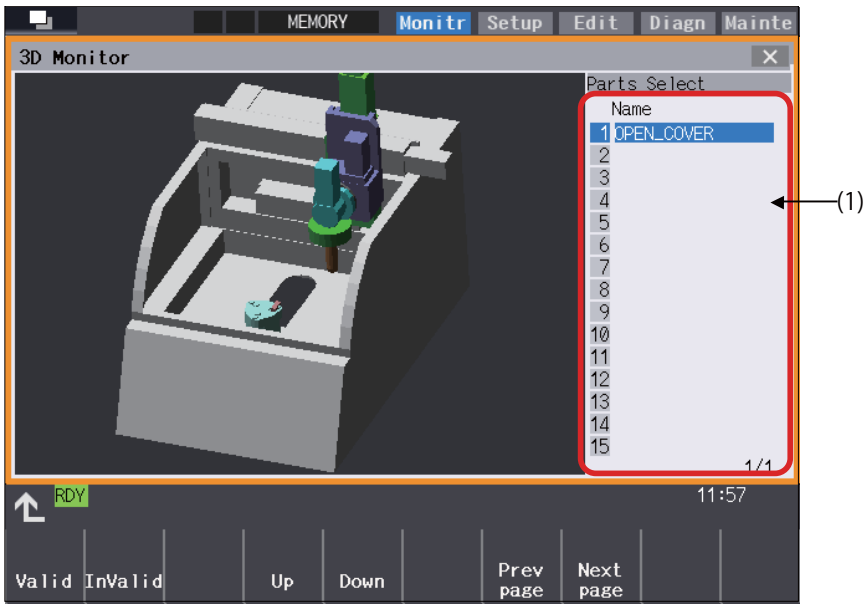
10.4.7.1 Parts Selection Screen

On the parts selection screen, you can switch whether or not to set the optional parts registered by MTB, as the interference check target.

The optional parts as the interference check target can be set to be shown/hidden on the display setting screen. For details, refer to "10.4.7.2 Display Setting Screen".

The optional parts as non-target of the interference check are not displayed in the 3D model display area.

Press the [Parts select] menu on the main screen to display the parts selection screen.



Display items

Display items	Details
(1) Parts list	The names of the parts that can be switched between target and non-target of the interference check are displayed. Select a part by the [Up]/[Down] menus, and press the [Valid] menu to set the part as the target. The name of the part set as the interference check target is highlighted. Up to 16 characters are displayed for the part name. Up to 99 parts can be displayed. The set condition remains after the power is turned ON again.

Menus

Menus	Details
Valid	This sets the selected part as the interference check target.
InValid	This excludes the selected part from the interference check target.
Up Down	These move the cursor position.

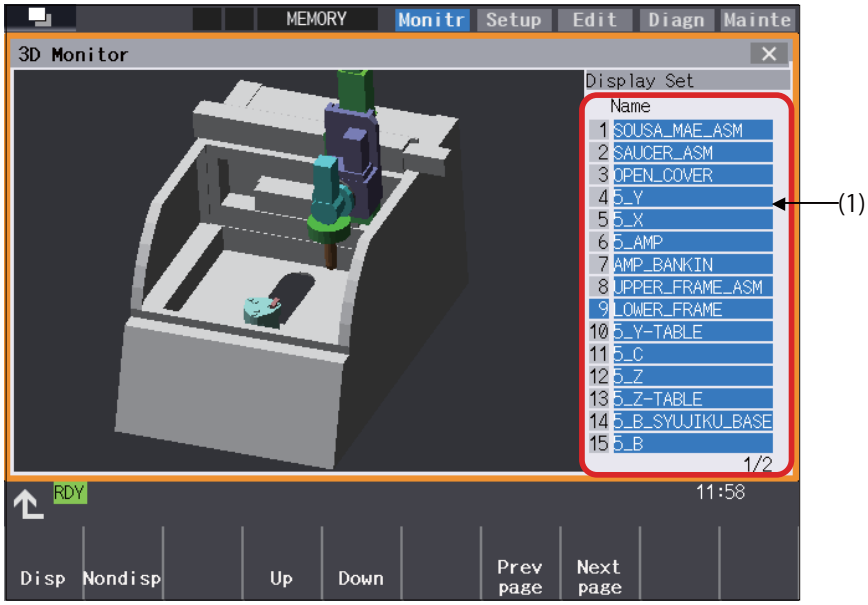
Menus	Details
Prev page Next page	These switch the display page when the parts cannot be displayed within one page.

10.4.7.2 Display Setting Screen

On the display setting screen, you can set whether or not to display each component of the machine model in the 3D model.

The interference check is performed regardless of whether or not the part is displayed.

Press the [Disp set] menu on the main screen to display the display setting screen.



Display items

Display items	Details
(1) List of display parts	The names of the parts that can be switched between show and hide are displayed. Select a part by the [Up]/[Down] menus, and press the [Disp] menu to display the part. The name of the part set as the display target is highlighted. Up to 16 characters are displayed for the part name. Up to 99 parts can be displayed. The set condition remains after the power is turned ON again.

Menus

Menus	Details
Disp	This sets the selected part as the display target. The target part is displayed in the 3D model display area.
NonDisp	This excludes the selected part from the display target. The target part in the 3D model display area is hidden.
Up Down	These move the cursor position.

Menus	Details
Prev page Next page	These switch the display page when the parts cannot be displayed within one page.

10.4.8 Inputting/outputting the Shape Group Data

The data of the registered shape group can be backed up. The backed-up data can be used in another machine.

Data of each shape group or all the shape groups in NC or memory card can be input/output.

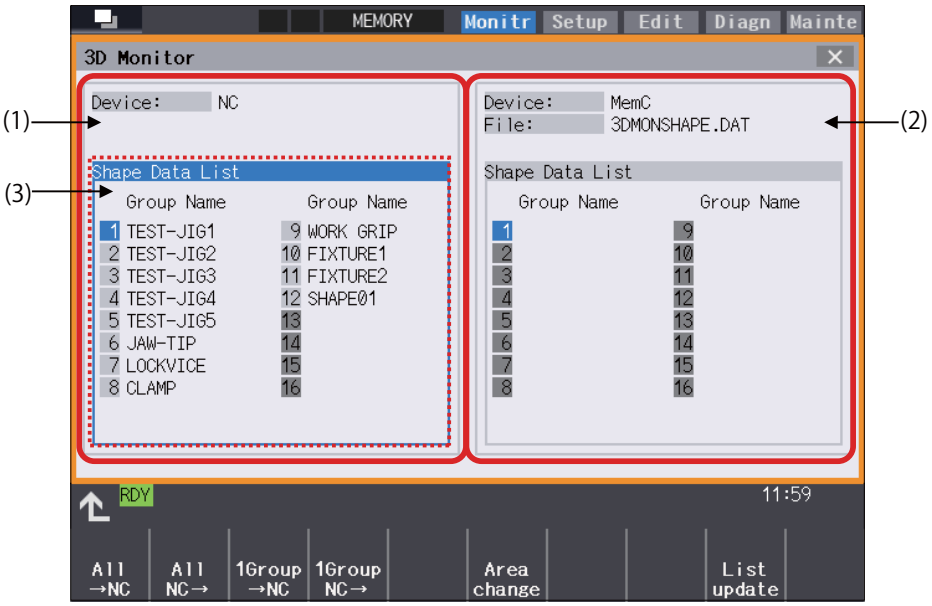
The file name of the shape group data is "3DMONSHAPE.DAT".

Note

(1) The file name cannot be changed.
(2) The data cannot be input from or output to any device other than a memory card.

The shape group is input/output on the data I/O screen.

Press the [Data I/O] menu on the main screen to display the I/O screen.



Display items

Display items		Details
(1)	Display area for files in NC	The list of the shape groups in NC to be input/output is displayed.
(2)	Display area for files in memory card	The file name and the shape group list in the memory card to be input/output are displayed.
(3)	Shape Data List	The group No. and group name of the shape data stored in each device are displayed. The display color of the group No. indicates the following state. Dark blue: Group No. that is currently selected Light gray: Group No. that can be selected Gray: Group No. where no data exists

Menus

Menus	Details
All →NC	This inputs the data of all the shape groups in the memory card, to NC.
All NC→	This outputs the data of all the shape groups in NC, to the memory card.
1Group →NC	This inputs the data of the group No. selected in the shape data list of the memory card, to the group No. selected in the shape data list of NC. When the shape group data exists in the group No. of the input destination, the overwrite confirmation message appears.
1Group NC→	This outputs the data of the group No. selected in the shape data list of NC, to the group No. selected in the shape data list of the memory card. When the shape group data exists in the group No. of the output destination, the overwrite confirmation message appears.
Area change	This switches the setting area between the shape data lists of NC and memory card. The active "Shape Data List" is highlighted in dark blue.
List update	This updates the group name in the shape data list of the memory card. When the memory card is removed or inserted while the data I/O screen is displayed, press this menu to update the shape data list.

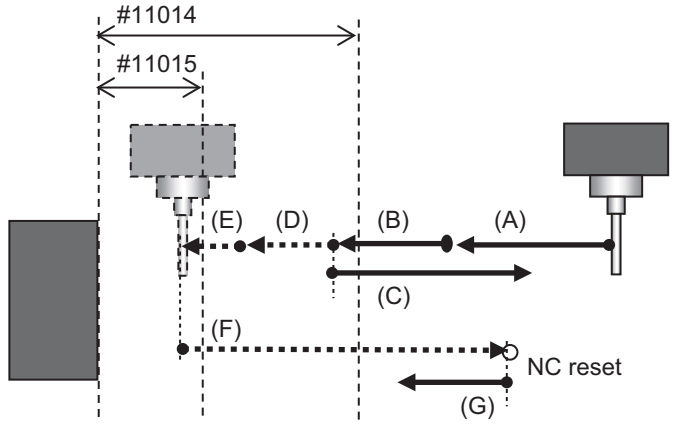
10.4.9 Machining the Workpiece During Interference Check

Interference check is performed at the machine position. For machine lock axis and axis during Axis detachment, the program position or the current position is different from actual machine position where the interference check is performed.

10.4.9.1 Manual Operation

During manual operation, the interference check is performed for the position where the axis will move next with the JOG or handle operation, in two steps using the interference check distances set by the parameters #11014 and #11015.

The operation is performed as follows. (A) to (G) in the sentences correspond to those in the drawing.



(1) The 1st-step interference check is performed in the initial state. The feedrate is clamped at the one determined with the distance of the parameter (#11014). (A)

(2) While the 1st-step interference check is performed, if the movement destination of the axis is within the distance of the parameter (#11014), the operation alarm "M01 Machine interference 1 0201" occurs. (B)

- JOG mode: The axis temporarily stops.
- Handle mode: The axis does not stop, the alarm is canceled immediately, and the 2nd-step interference check is performed.

After the alarm occurs, if the axis moves away from the interfering object, the alarm is canceled, and the 1st-step interference check is continuously performed. (C)
After the alarm occurs, if the axis moves more closer to the interfering object, the alarm is canceled, and the 2nd-step interference check is performed. The feedrate is clamped at the one determined with the distance of the parameter (#11015). (D)

(3) While the 2nd-step interference check is performed, if the movement destination of the axis is within the distance of the parameter (#11015), the operation alarm "M01 Machine interference 1 0202" occurs, and the axis stops. (E)

When the alarm occurs, the axis can move only away from the interfering object.
After the alarm occurs, even if the distance from the interfering object is further than the distance of the parameter (#11014), the 2nd-step interference check is continuously performed. (F)

(4) The alarm is canceled with NC reset, and the state returns to the initial state (1st-step interference check). (G)

10.4.9.2 Automatic Operation

During automatic operation, the interference check is performed in one step using the 1st-step interference check distance set with the parameter #11014. The check does not proceed to the 2nd step.

When the distance between the movement destination of the axis and the interfering object is shorter than the distance set with the parameter (#11014), the operation alarm "M01 Machine interference 1 0201" occurs, and the axis stops.

10.4.9.3 Canceling the Machine Interference Alarm

When the machine interference alarm occurs, the alarm can be canceled by one of the following ways.

(1) Move the axis manually away from the interfering object.

(2) Execute NC reset.

(3) Turn OFF the [Colli check] menu on the main screen to disable the machine interference check.

10.4.9.4 Disabling the Interference Check

In the following cases, the operation alarm "M01 Interference check disabled 0200" appears and the interference check is not performed.

(1) When the [Colli check] menu on the main screen is OFF

(2) When there is any axis that has not returned to the reference position (The axes during Spindle position control (Spindle/C axis) and the axes during Axis detachment are excluded.)

(3) During skip retract

(4) During arbitrary reverse run

(5) During manual speed command

(6) During manual arbitrary reverse run

(7) When the machine model is invalid or when there is no machine model

(8) When spline interpolation2 is enabled (including during temporary cancel)

(9) When the top of the machining program is any of the following commands and single block stop, feed hold stop or re-calculation signal is turned ON while the block is executed

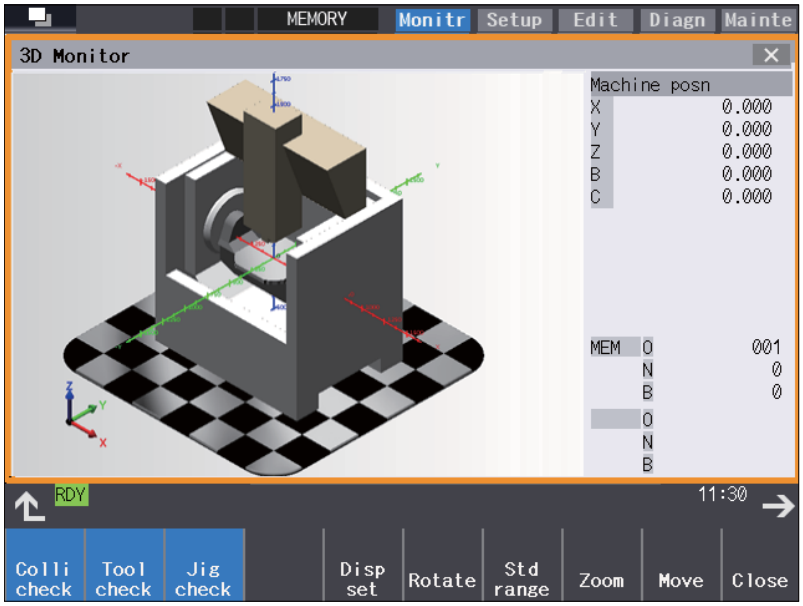
- Reference position return (G28/G30) (*1)
- Start position return (G29)
- Unidirectional positioning (G60)

(*1) To command it at the top of the MDI program, turn OFF the menu [Colli check] on the 3D monitor screen beforehand. If it is commanded while the menu is ON, it may cause an alarm such as "Z82 Interference check error 0004", "M01 Machine interference 1 0201", and so on at a movement after reference position return.

10.5 Real-time 3D Machine Interference Check (M System) [M800VW Series]



This function checks the machine operation in real time by using 3D model of machine, tool and jig. The movement destination of the machine is checked in advance by using the 3D model, an alarm is displayed as soon as an interference is detected, and all the axes are decelerated to stop. A translucent 3D model can be displayed at the pre-read position.



For details on operations, etc. of this function, refer to "10.4 3D Machine Interference Check (M System) [M800VW/M80VW Series]".

10.6 Interference Check III

Monitr — 

(For 15-type/19-type display unit)

Monitr — 

(For 8.4-type/10.4-type display unit)

This function checks the relative relation between interfering objects to prevent their interference. On the interference check III screen, 3D models of the selected interfering objects are displayed.

When an alarm occurs, the target interfering object is displayed in yellow or red depending on the alarm type.

M03 Interference detected: Red
M03 Entry in interfere alarm area: Red
M03 Entry in interfere warn area: Yellow

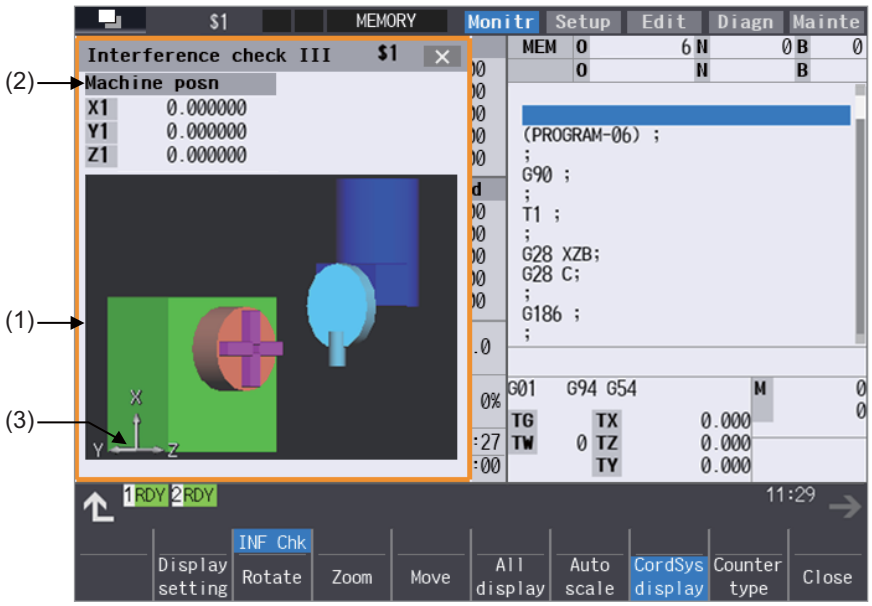
The interfering objects disabled by the system variable or R register "Interfering object enable/disable designation" are not displayed on this screen.

The 3D models of the interfering objects displayed on the screen are updated at the following timings.

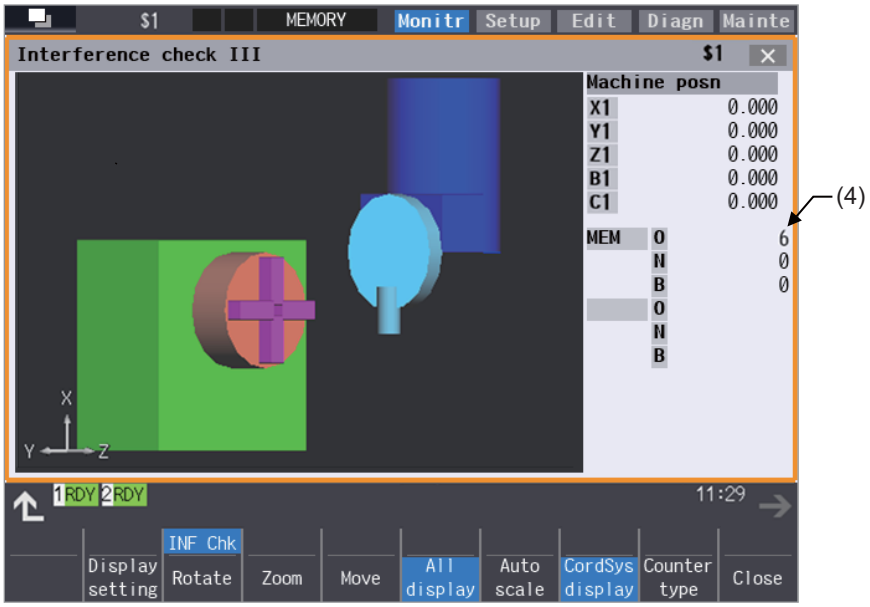
- When the interference check III screen is opened
- When the interfering object selection is enabled by "Enable interfering object selection data" while the interference check III screen is displayed

10.6.1 Main Screen

■Normal display ([All display] is OFF)



■Full-screen display ([All display] is ON)



Display items

Display items		Details
(1)	3D model display area	The 3D models of the selected interfering objects are displayed according to the indication of the machine position. The 3D models move to the position according to the machine position even not during the interfering check III mode. The 3D models are displayed in the color set by the solid/interfering object definition file (OBSTACLE.DAT). The color (RGB value) may be displayed in an approximate value depending on the display unit specifications.
(2)	Counter	The selected counter is displayed. In the normal display mode, the axis names common to part systems ("#1022 axname2") based on the basic axis name set by the parameters "#1026 base_I", "#1027 base_J", and "#1028 base_K" are displayed. In the full-screen display mode, maximum 5 axes are displayed. Three axes are set by #1026, #1027 and #1028, and two axes are set by the rotary axis configuration parameters. When the rotary axis configuration parameters are not set, two rotary axes set by the parameter "#1013 axname" are sequentially displayed. The counter to be displayed can be selected by the menu [Counter type].
(3)	Coordinate system display	This displays the coordinate system direction. This displays the axis direction of the base axis I/J/K with arrow.
(4)	Program No. display	In the full-screen display mode, the currently executed program number, sequence number, and block number are displayed. While a subprogram is executed, its program number, sequence number, and block number are displayed.

Menus

Menus	Details
Disp set	This sets whether to show/hide the selected interfering object.
Rotate	This rotates the 3D model. It is rotated around the center coordinate of all selected interfering objects in the specified direction.
Zoom	This enlarges/reduces the 3D model. It is enlarged/reduced by 5% based on the center coordinate of the 3D model display area.
Move	This moves the 3D model. It is moved by 20 pixels from the current display position in the specified direction.
All display	This switches the normal display mode and the full-screen display mode.
Auto scale	This enlarges/reduces the entire 3D models to fit the drawing area, and initializes the rotation.
CordSys-display	This changes whether to show/hide the coordinate system direction in the 3D model display area. The coordinate system direction is held after the screen is switched, but it is not held after the power is turned OFF and ON again.
Counter type	This changes the counter type to be displayed on the interference check III screen. The selected counter type is held after the power is turned OFF and ON again. The counter type is common between part systems.
Close	This closes the interference check III screen to return to the monitor screen.

Sub-menus of [Rotate]

Menus	Details
Up	This rotates the 3D model in the vertical/horizontal directions. The same operation can be performed by pressing the cursor keys on the NC keyboard.
Down	
Left	
Right	
CCW	This rotates the 3D model counterclockwise.
CW	This rotates the 3D model clockwise.
Rev Dsp	This flips the 3D model 180 degrees horizontally.
Rotate Init	This initializes the rotation of the 3D model display.

Sub-menus of [Zoom]

Menus	Details
Zoom In	This enlarges/reduces the 3D model. The same operation can be performed by pressing the [+]/[-] keys on the NC keyboard.
Zoom Out	

Sub-menus of [Move]

Menus	Details
Up	This moves the 3D model in parallel vertically and horizontally. The same operation can be performed by pressing the cursor keys on the NC keyboard.
Down	
Left	
Right	

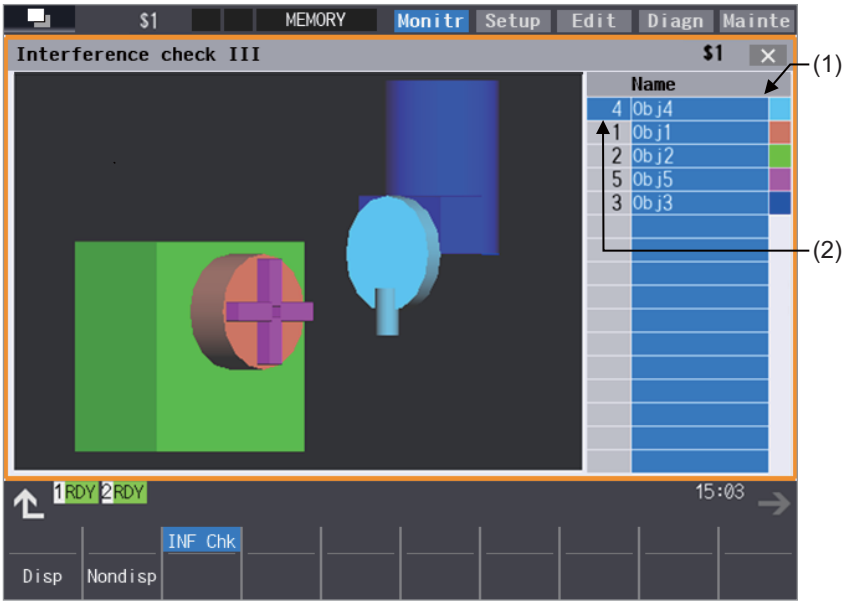
Sub-menus of [Counter type]

Menus	Details
Relat posn	The counter to be displayed in the interference check III screen can be set by pressing a sub-menu of [Counter type]. After the counter is set, the screen returns to the interference check III screen. <Note> ♦The menu [Wk Comp co posn] is grayed out and non-selectable when the workpiece installation error compensation specification is not valid. ♦The menu [Incline co posn] is grayed out and non-selectable when all the specifications of inclined surface machining, simple inclined surface machining and R-Navi are not valid.
Work co posn	
Machine posn	
Remain command	
Next command	
Manual int amt	
Program posn	
Tip wk posn	
Tip machine	
Pulse	
Table co posn	
Wk Comp co posn	
Incline co posn	

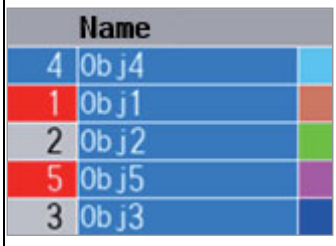
10.6.2 Display Setting Screen

On the display setting screen, you can set whether or not to display the interfering object. The interfering object can be hidden in order to identify the interference part easily when an interference check III alarm occurs.
The interfering object is switched to be shown or hidden by selecting the interference object from the list and pressing the menu [Disp]/[Nondisp]. The interfering object set as hidden is not displayed in the 3D model display area; however, the interference check is performed.

Press the [Display setting] menu on the main screen to display the display setting screen.



Display items

Display items		Details
(1)	Interfering object list	Interfering object definition No., name and color of the selected interfering object are displayed in the order from the 1st to the 16th interfering objects. The name of the target interfering object is highlighted. The set state of show/hide is held after the power is turned OFF and ON again. When an interference check III alarm occurs, the target interfering object definition No. field is displayed in red or yellow in the same way as the 3D model display. (Example) The operation alarm "M03 Interference detected" occurs in the interfering object definition Nos. 1 and 5. 
(2)	Cursor	It moves by the menus [↑]/[↓] to select the interfering object to be shown/hidden.

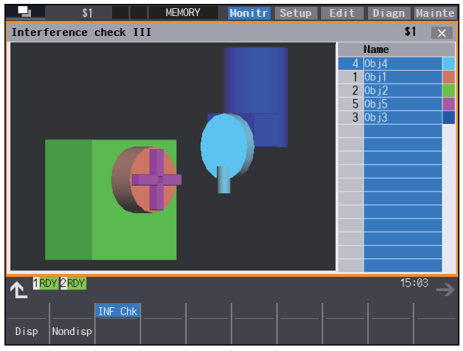
Menus

Menus	Details
Disp	This sets the selected interfering object as the display target to highlight the interfering object name. The target interfering object is displayed in the 3D model display area.
NonDisp	This sets the selected interfering object as the non-display target to remove the highlight from the interfering object name. The target interfering object is hidden from the 3D model display area.

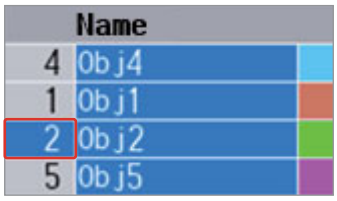
10.6.3 Operation Method

Changing whether to show or hide the interfering object

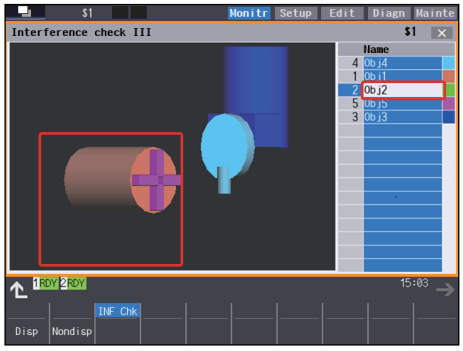
(1) Press the menu [Display setting]. → The display setting screen is displayed.



(2) Press the cursor movement keys [↑]/[↓] to select the interfering object to be shown/hidden. → The selected interfering object No. field is highlighted.



(3) Press the menu [Nondisp]. → The highlight on the target interfering object name is removed, and the target interfering object is hidden from the 3D model display area.



The show/hide state of the interfering object is held as the screen setting. Even when another interfering object is selected, the show/hide state is not changed. The show/hide state can also be set for the row of unselected interfering object.

Rotating the 3D model display

(1) Press the menu [Rotate].
(2) To rotate by 5 degrees: Press the menus [↑]/[↓]/[←]/[→] or cursor movement keys to adjust the display angle.
To rotate by 180 degrees: Press the menu [Rev Dsp].

Enlarging/reducing the 3D model display

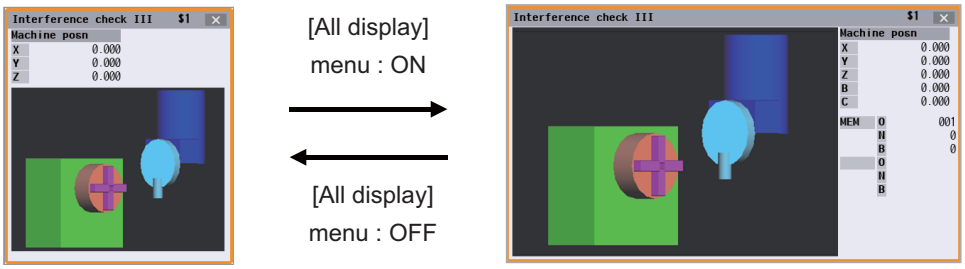
(1) Press the menu [Zoom].
(2) To enlarge the 3D model: Press the menu [Zoom In] or [-] key.
To reduce the 3D model: Press the menu [Zoom Out] or [+] key.

Moving the 3D model display

(1) Press the menu [Move].
(2) Press the menu [↑]/[↓]/[←]/[→] or cursor movement keys to adjust the display position.

Switching the full-screen display mode

Press the menu [All display] to display the interference check III screen on the whole screen (full-screen display mode). Press this menu again to return the normal display. The selected display mode is maintained even after rebooting the machine.



10.7 Restart Search

Monitr – 

With this function, the program search can be executed while updating the coordinate values and the modal information in the same manner as an actual program operation.
Therefore, this function enables machining from a block in the middle of the program as if the actual machining was carried out from the beginning.
This function is used to restart machining when a machining program is to be resumed after it has been suspended midway due to tool damage or for some other reason.

When the machining is suspended due to tool damage or power interruption, the line No. of the machining program is kept in CNC. Restart search can be executed by specifying this line No.

Note

(1) The line No. kept in CNC is not always the same as the line No. that can be checked in the text editor on PC or the line No. counted on the Monitor screen. Therefore, the line No. displayed on the Restart search screen needs to be checked.

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 No./block No. or line No. 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> ♦When NC is reset or the power is turned OFF while the interruption of fixed cycle, special fixed cycle, MTB macro, or macro is executed, the search without inputting anything in the input section will find each source program. ♦When the user macro is executed, the user macro execution position is searched.	
Restart type 2	If a machining program different from the machining program to be restarted was run before starting restart search, specify the sequence No./block No. or line No. before restarting the machining.	
	Automatic top search OFF	A top search must be executed from the screen. Then, command a sequence No./block No. or line 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 program No./sequence No./block No. or program No./line No., and restart the 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".

There are the following two designation methods of restart.

■Sequence No. designation method

Specify the program position of restart search with the program No./sequence No./block No./repeat count. (Specify the ONBP numbers.)
The machining can be restarted by specifying the sequence No. and block No. of the machining program.

■Line No. designation method

Specify the program position of restart search with the program No./line No./repeat count. (Specify the OLNP numbers.)
The position to restart the machining can be specified with the line No. even in the machining program where the sequence No. is duplicated.
When the menu [Specif.LineNo.] is highlighted, this method is used.

In two or more part systems, when all the following conditions are met, the restart search can be executed for all part systems in batch.

If any of the following conditions is not met, such as that restart type 2 is selected, the restart search is executed for each part system by changing the displayed part system.

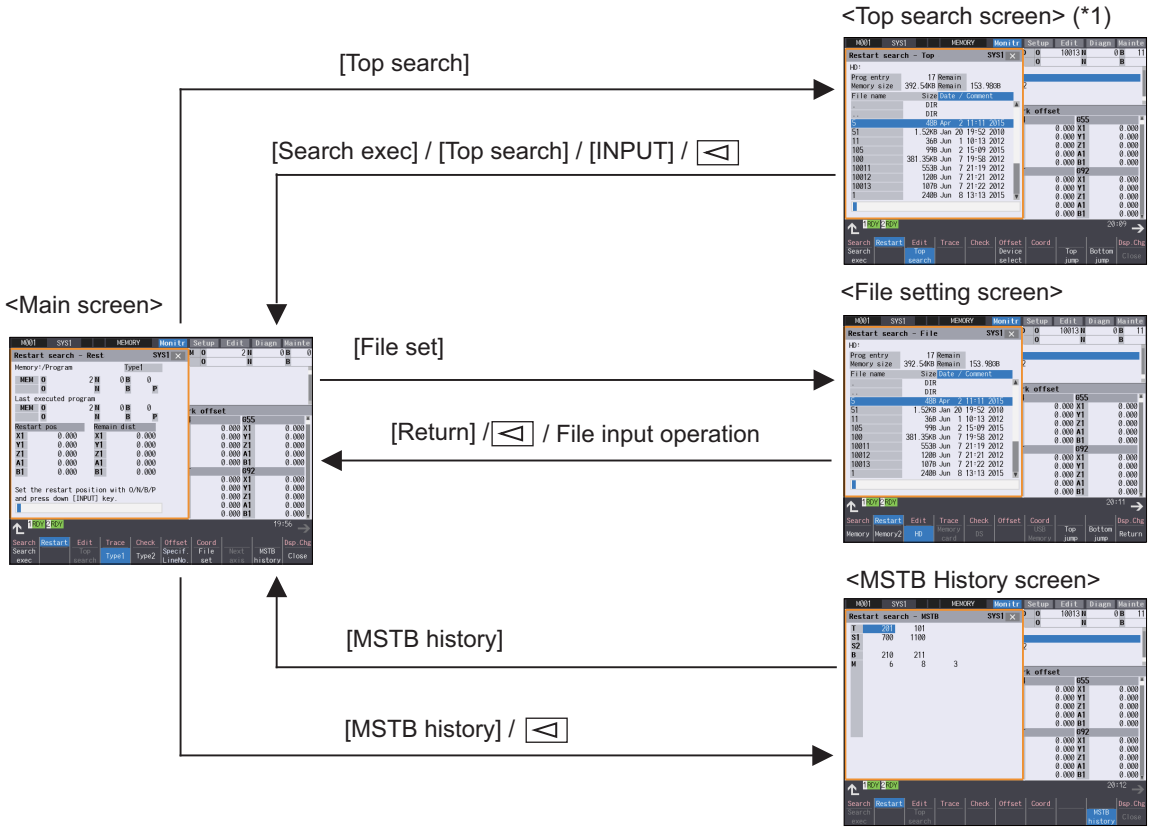
<Conditions of restart search for all part systems in batch>

- The parameter "#1285 ext21/bit0" is set to "1" (Multi-part system program management is enabled.)
- The parameter "#1285 ext21/bit1" is set to "1" (Search through a program for all part systems at once is enabled.)
- In all the part systems, programs of the same number in NC memory or NC memory 2 are searched for.
- Restart type 1 is selected.
- Restart search is performed with the sequence No. designation method.

Note

(1) Even if the above conditions are met, when the parameter "#8141 Restart srch ONBP" is set to "0" and any of the ONBP number is designated, the restart search is executed for each part system. When the ONBP number is not designated, the restart search is executed for all part systems in batch.

Screen transition



(*1) 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 search)

Precautions

(1) Set the tool offset amounts and parameters before proceeding the program restart search. If they are not set beforehand, the axes will not return to the proper machining start position.

(2) The axes will not return to the proper machining start position if a program using the user macro external signal input, machine coordinate readout or external mirror image command executes the restart search.

(3) If an attempt was made to shift the coordinate systems by manual or MDI interrupt the last time the machining program was being executed, the axes will not return to the proper machining position.

(4) When using the type 1, a top search for the head of the program is not available. Restart search of the type 1 searches from the previous operation start block and locates the designated block as it memorizes the operation start block previously operated.

(5) When "#8914 Auto Top search" is set to "0" for the type 2, an error message "Top search not completed" appears if the search is attempted directly with the type 2 without executing the top search. Conduct the top search first.

(6) The macro statement blocks cannot be searched for both type 1 and type 2. It is possible to execute the restart search if the control parameter "#8101 MACRO SINGLE" is set to ON. However, note that the tool path may change because of the radius compensation, the corner R/C, and geometric read ahead blocks.

(7) When a comment line, macro line, control statement , etc. is specified in restart search operation with the line No. designation method, the search is completed at the previous block. When the specified line is exceeded due to a control statement such as GOTO, etc., the search is completed at the previous block.

(8) When executing the restart for the type 1 with a tape, the axes will not return to the proper machining start position if the operation starts from midway of the tape.

(9) WHILE/GOTO statement cannot be used for a tape operation. Thus, if a restart search is executed with the tape operation which includes WHILE/GOTO statement, a program error (P295) will occur.

(10) When the linear-type rotary axis returns to the restart position, control returns to the workpiece coordinate position.

(11) A program restart search is not available at the head of the program. Restart search settings are possible, but control may not return to the correct position when the head of the program is searched.

(12) If the parameter "#1151 rstini" is set to "0" (No reset initial) in the program without the sequence number, a restart search (type 1) may not function properly after reset 1 has been executed. Therefore, assign a sequence number to each program.

(13) If INPUT/search is carried out without designating any value in the input area, a restart search is performed using the previously executed ONB number (type 1 only). If a reset is executed during MDI interruption processing, a restart search is performed using the previous ONB number for MDI interruption.

(14) If INPUT/search is carried out without designating any value in the input area, the position of the ONB number that first matches the designated condition is searched at restart, regardless of the number of repetitions.

(15) When the previously executed program is RS232C, the operation message "Search error" appears if INPUT/search is carried out without designating any value in the input area.

(16) If a restart search is carried out while an error remains in the program, it may cause a search error. When a search error occurs, correct the program after resetting, and perform a restart search again.

(17) When the operation mode is set to the MDI mode, a restart search is not available. If an attempt is made to carry out a restart search, the operation message "Can't search in MDI mode" is displayed.

(18) Restart search is available for a program that has a file name consisting of up to 32 characters. Restart search is not available for a program that has a file name consisting of 33 or more characters.

(19) Restart search is not available for sub part system I/II for M80V Series. If restart search is carried out, the message "Search error" appears.

(20) Restart search is not available for sub part system II for M800V Series. If restart search is carried out, the message "Search error" appears.

(21) A search error will occur if an intended ONB is not found even M99 block is detected in the main program.

(22) An error will occur if the M99 block in the main program is searched.

(23) When "#1122 pglk_c" (Program display lock C) is set to "2", the restart search cannot be executed as "Program display lock C" is applied.

(24) If both N and B are omitted, or if the line No. is omitted, the search is performed, assuming that the head of the program and the repeat count "1" are specified.

(25) When either N or B is omitted, the omitted one is assumed as "0".

(26) Restart search cannot be carried out during the surface selection by R-Navi. A program error (P49) will occur.

(27) Including the line No. designation method, restart search cannot be carried out for a block that comes after any of the following command blocks. A program error (P49) will occur.
3-dimensional circular interpolation, cylindrical interpolation, polar coordinate interpolation, milling interpolation, tool center point control, simple tool center point control, inclined surface machining, simple inclined surface machining, workpiece installation error compensation, axis name switch, mixed control, machining condition selection I, sub part system control I, sub part system control II, direct command mode, or arbitrary axis exchange control

(28) When you execute the restart search to a block in synchronous tapping cycle modal, issue the S code for a tapping spindle from MSTB history menu. If it is not issued, the tapping spindle will execute the synchronous tapping with current gear step without changing the gear.

(29) The number of program repetitions, "P", can be designated up to the total value of the same number of sub-program repetitions, "L", described on the program.

(30) When "#8129 Subpro No. select" is set to "1" or "2", it triggers a restart search on a sub-program with the O number. Therefore, to designate the program number, it must be prefixed by an "O".

(31) When an operation search or type 2 top search is carried out, the last program which is operating immediately before the search is retained. However, type 1 restart search cannot be carried out for the program. If type 1 restart search is carried out in this situation, the message "Search error" displayed. Perform type 2 restart search.

(32) When a part system under automatic operation, program restarting, or program checking is searched, the message "Executing automatic operation", "Program restarting", or "Program checking" is displayed.

Precautions (For multi-part system)

(1) For a program where the timing synchronization command is used, do not execute restart search in the block with the timing synchronization command and the subsequent blocks. The block execution timing is off between the part systems, and the machine is in danger of collision.

■Restart search for all part systems in batch

(1) When the restart search is completed, the number of the searched part system is displayed in the format "$ + part system number" following the message "Search completed".

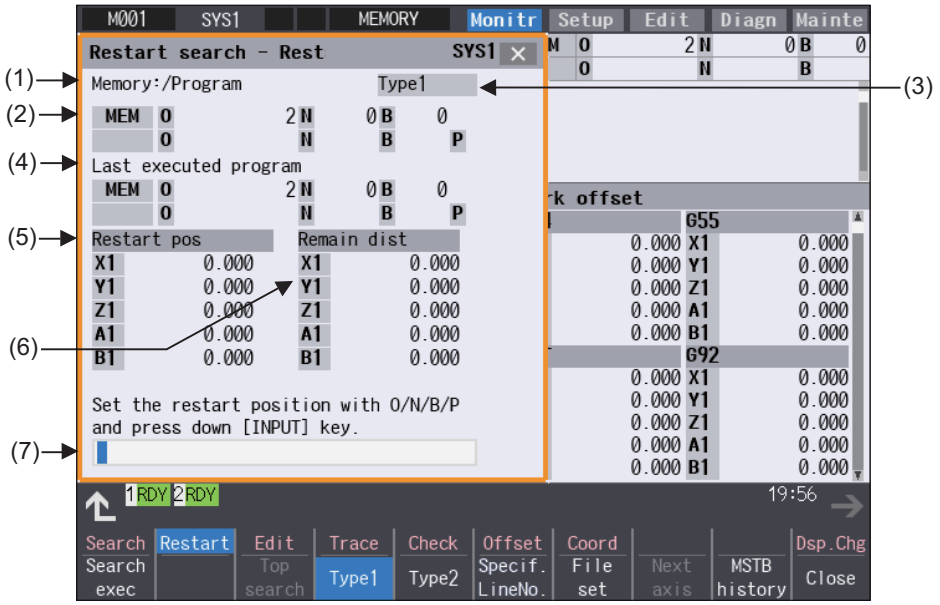
(2) When the restart search is executed for all part systems in batch in a system with sub part system, only the main part system is searched. (The sub part system is not searched.)

(3) When the parameter "#8141 Restart srch ONBP" is set to "1" and the restart search is executed in individual part systems, select restart type 2.

(4) When a search error occurs, the numbers of all the part systems that are judged to be abnormal are displayed in the format "$ + part system number" following the message "Search error".

10.7.1 Main Screen

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



Display items

Display items		Details
(1)	Device name, 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. (*1), repeat count, block No.).
(3)	Restart type	This displays the restart search type.
(4)	Last executed program position	This displays the last executed program position (program No., sequence No., block No., repeat count). When a restart search is executed with restart type1, actual program restart position may not match the position of last executed program. Because the position is updated at the timing of machine stop operations (NC reset, single block stop or feed hold stop).
(5)	Position when restart search is completed	This displays the position on the local coordinate system when the restart search is completed.
(6)	Remaining distance when restart search is completed	This displays the remaining distance when the restart search is completed.
(7)	Input section	This displays details of the key input.

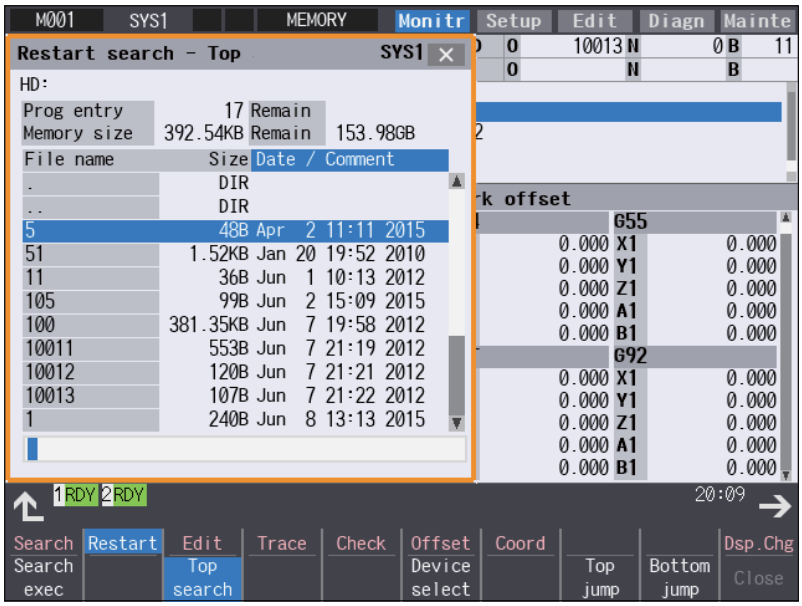
(*1) If the sequence number exceeds six digits, only the low-order six digits are displayed.

Menus

Menus	Details
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).
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 to "1", this menu cannot be selected.
Type1	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".
Type2	
Specif. LineNo.	This switches the No. designation method of restart search. When this menu is highlighted, the line No. designation method is selected. When it is not highlighted, the sequence No. designation method is selected.
File set	This changes to the pop-up window for setting the file, and enables the file to be selected.
Next axis	This switches the axis to be displayed on the counter with 8.4-type or 10.4-type. (*1)
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] key is pressed, that command will be executed.
Close	This closes the pop-up window and quits this function.

(*1) The menu will be grayed out and cannot be selected when the number of valid axes is six or less.

10.7.2 Top Search Screen



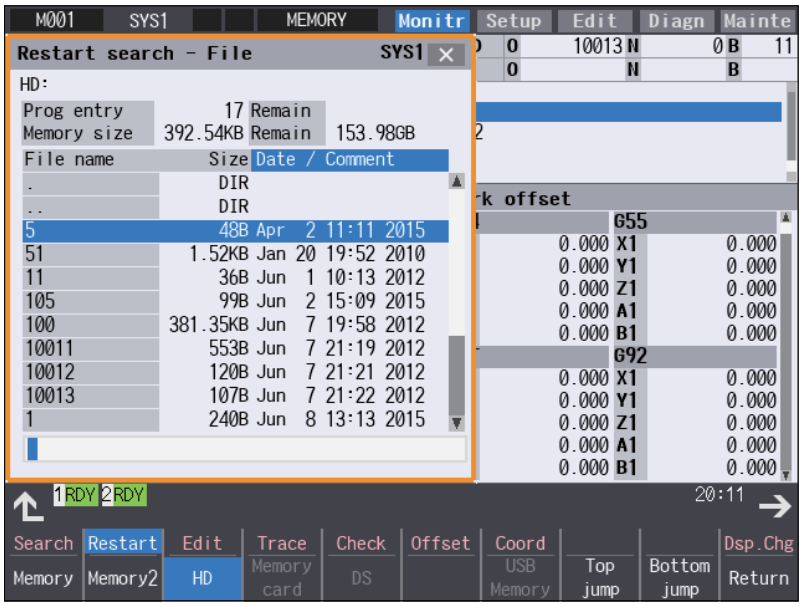
Display items

Same as "Operation Search".

Menus

Menus	Details
Search exec	This starts the top search based on the designated device, directory, program number (O), sequence number (N), block number (B) and number of block execution times (P).
Top search	When the menu 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.
Device select	This displays the device menu. Select which device program to search for from this menu. When a device with directory is selected, the root is selected first.
Top jump	Display starts from the first page of the directory/list of files. The cursor moves to the top line.
Bottom jump	The last page of the directory/list of files is displayed. The cursor moves to the bottom line.
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.
Preview	This switches to show or hide the preview (detailed information).
Change Disp	This changes the display format of the file/directory list.
List update	This updates the list contents. (The latest contents of the currently selected device and directory are listed.)
Sort change	This changes the method of sorting the list.

10.7.3 File Setting Screen



Display items

Same as "Operation Search".

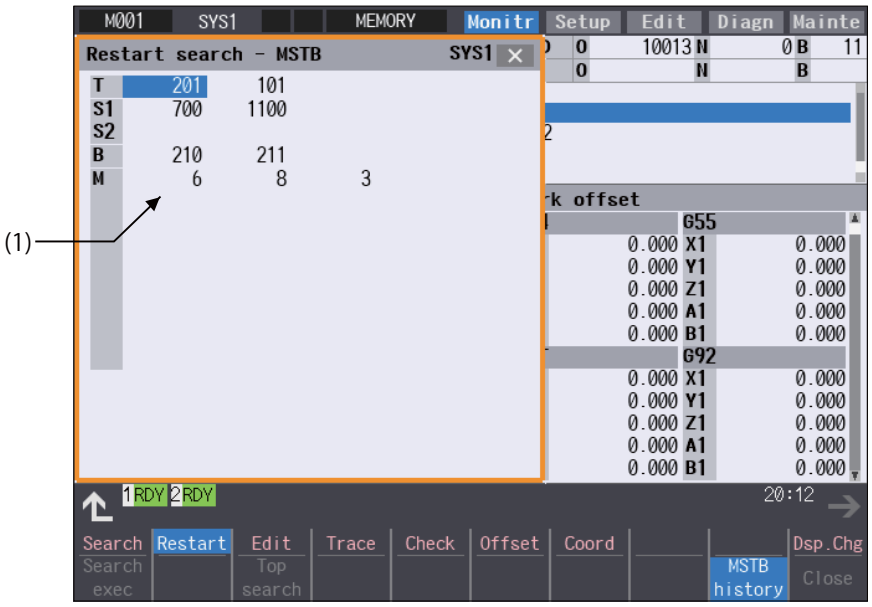
Menus

Menus	Details
Memory Memory2 HD Memory card DS USB Memory	This selects the device for searching for the program. When a device with directory is selected, the root is selected first. This menu [HD] is not displayed on M800VS/M80V Series. (*1)
Top jump	Display starts from the first page of the directory/list of files. The cursor moves to the top line.
Bottom jump	The last page of the directory/list of files is displayed. The cursor moves to the bottom line.
Return	This returns the main menu.
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.
Preview	This switches to show or hide the preview (detailed information).

Menus	Details
Change Disp	This changes the display format of the file/directory list.
List update	This updates the list contents. (The latest contents of the currently selected device and directory are listed.)
Sort change	This changes the method of sorting the list.

(*1) When the HD mode operation on an industrial computer (IPC) is enabled with M80V Series, the menu [HD] becomes selectable and you can select files in the hard disk on the industrial computer. The availability of this function depends on the MTB specifications (parameter "#1760 cfgPR10/bit2").
Set the IP address of the industrial computer to the parameter #11005 to use this function.

10.7.4 MSTB History Screen



Display items

Display items	Details
(1) MSTB history	The MSTB codes used for the machining are listed after the restart search is completed. The displayed value can be set (manual numerical value command) as the current modal value until the reset or starting is applied. The character color is changed for the value which is once set. The number of displayed codes is as follows: M: up to 35, S1 to S8: 3 each, T: 3, B:3. If the number of codes used for the machining exceeds the displayable number of codes, the codes which have been used in the beginning is not displayed. Address T will display only the tool number regardless of the parameter (#11038 T disp type).

Menus

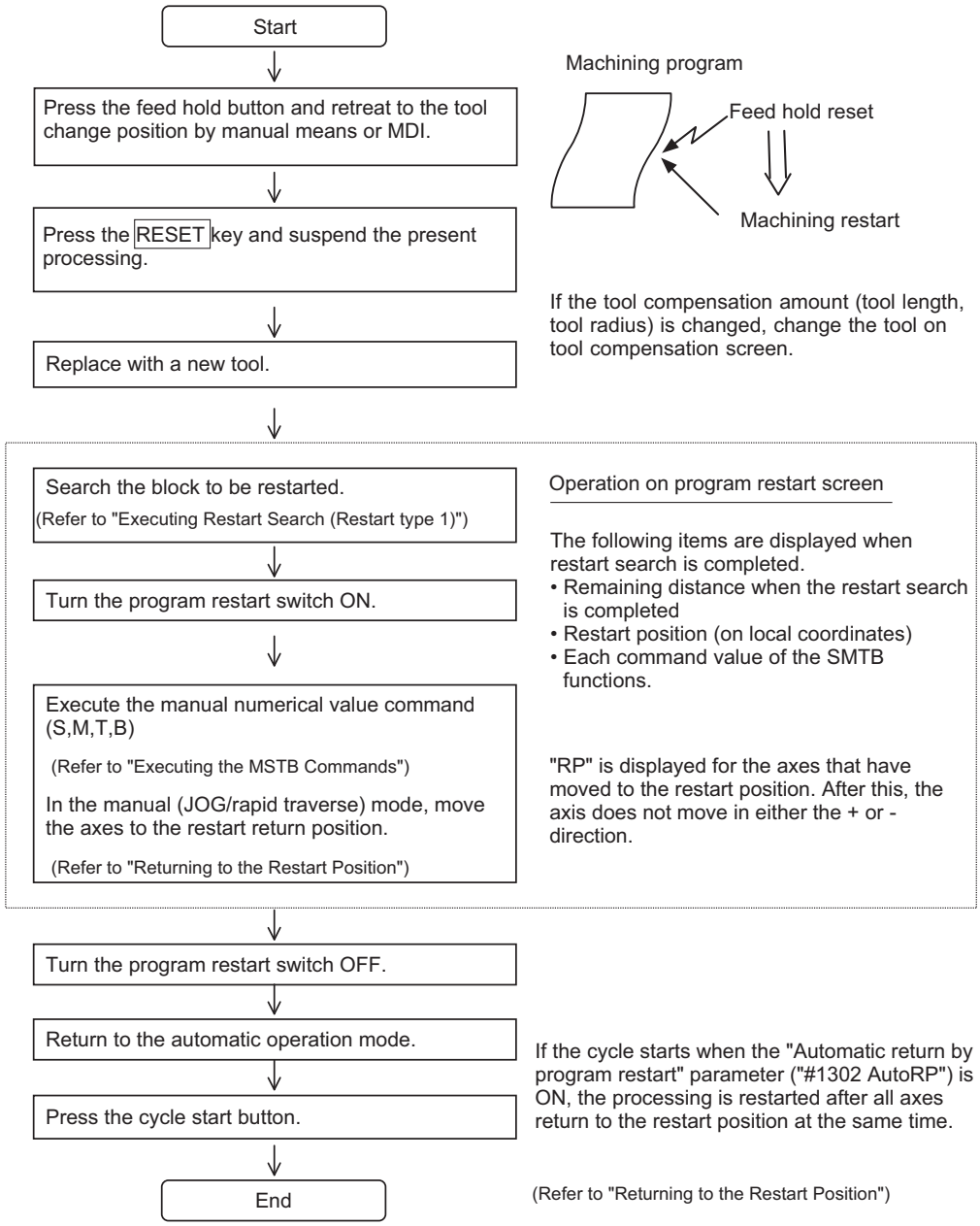
Menus	Details
MSTB history	When this is highlighted, it indicates that the MSTB history pop-up window is open. If the menu is pressed when this menu is highlighted, the system will close the MSTB history pop-up window, and the main pop-up window opens.

10.7.5 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.



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.)

(2) Execute "reset & rewind" for resetting.

(3) Restart search is not available for a program that has a file name consisting of 33 or more characters.

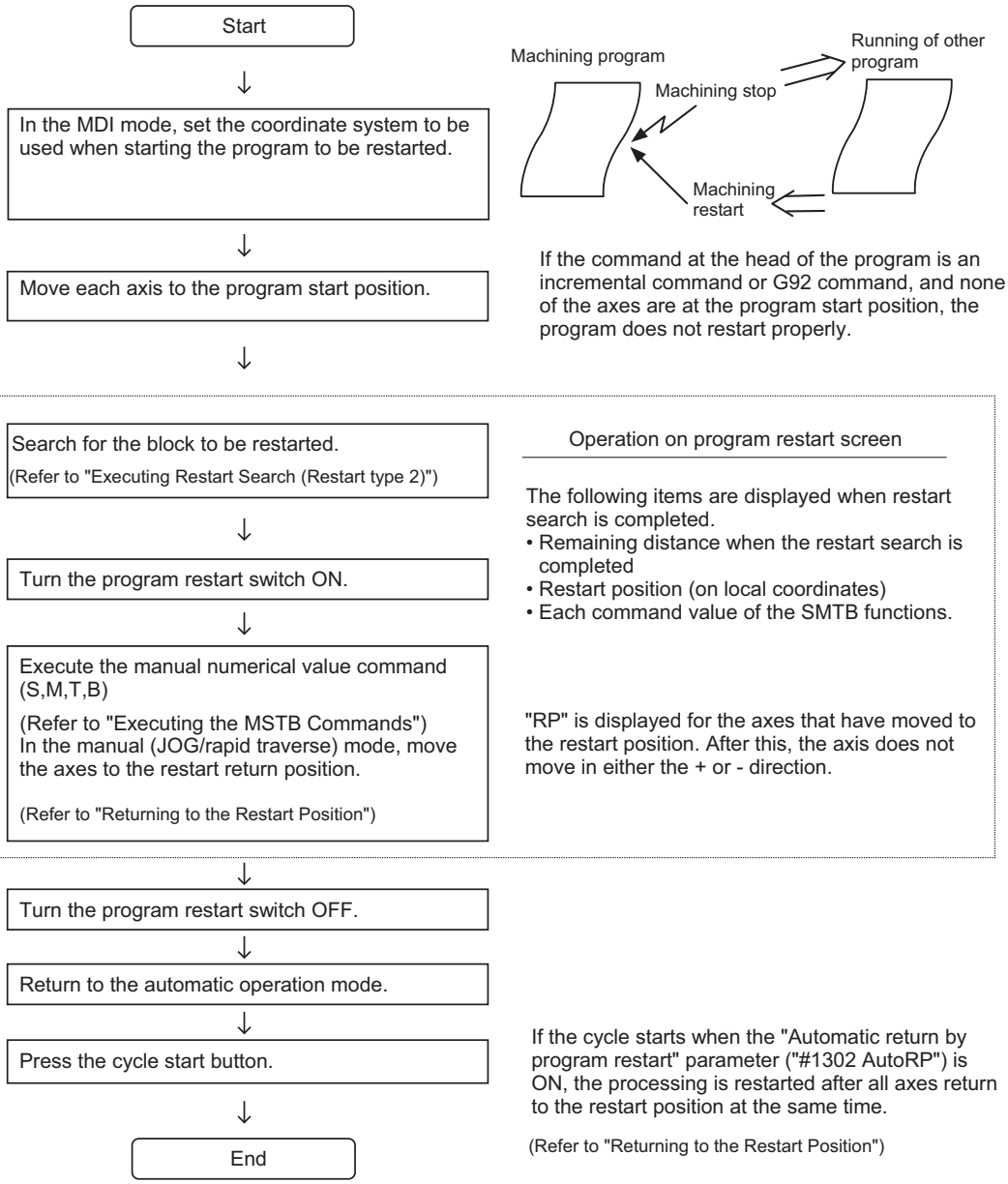
Restart type 2

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

(1) 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.)

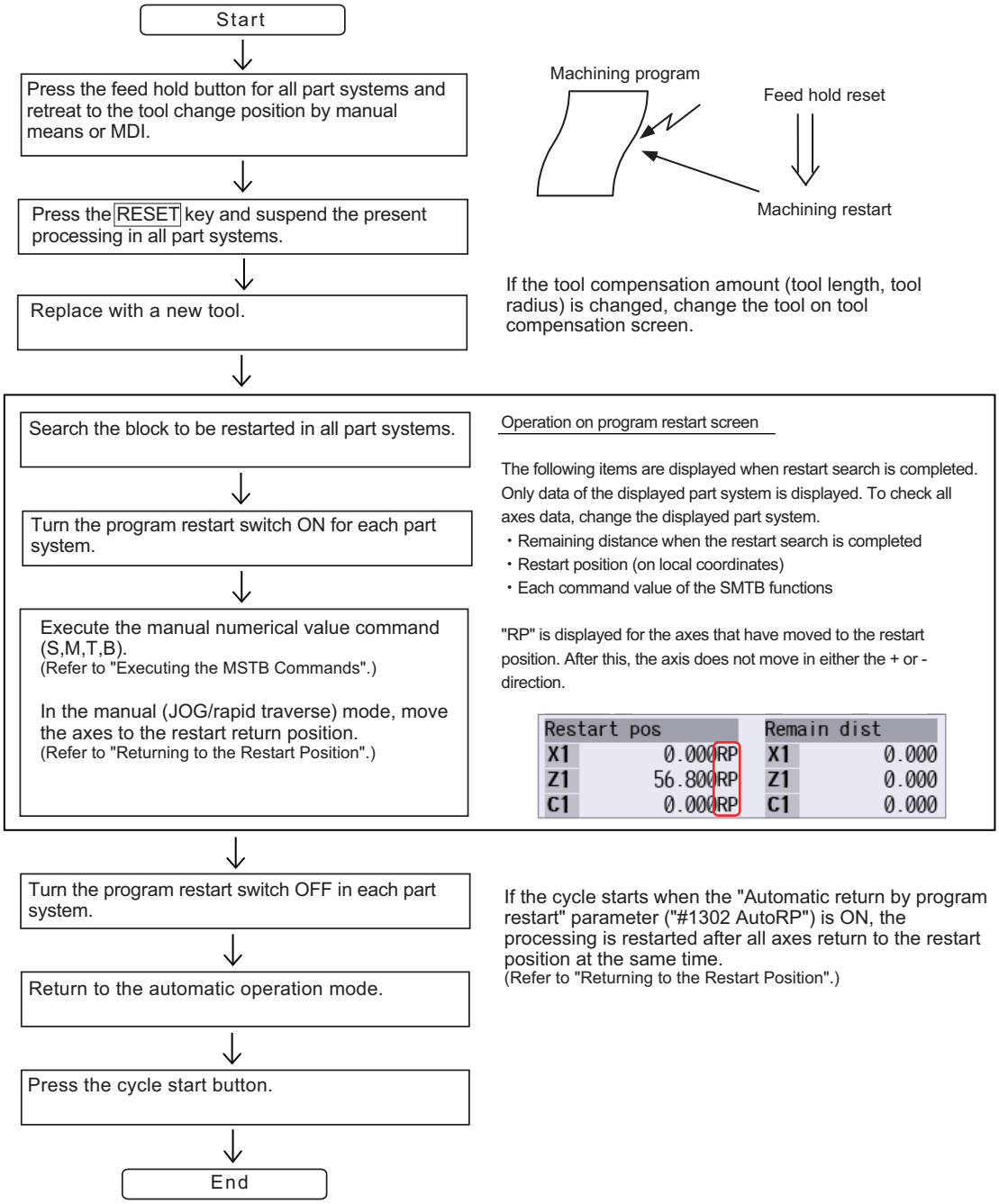
(2) Restart search is not available for a program that has a file name consisting of 33 or more characters.

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

Restart search for all part systems in batch

When feed hold has been applied and reset in all the part systems because the tool has broken, etc., the restart search is executed for all part systems in batch.

For conditions of restart search for all part systems in batch, refer to "10.7 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.)

(2) Execute "reset & rewind" in all part systems for resetting.

(3) Restart search is not available for a program that has a file name consisting of 33 or more characters.

10.7.6 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 following conditions are met, the program search is executed across all part systems in batch.

- The parameter "#1285 ext21/bit0" is set to "1" (Multi-part system program management is valid).
- The parameter "#1285 ext21/bit1" is set to "1" (Search through a program for all part systems at once).
- The machining program is stored in the device [Memory].
- ONBP numbers are not designated.

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.
When using tape operation, index the top of the tape.

(3) Press the main menu [Restart]. → The sub-menu is displayed.
The main screen for restart search appears.

(4) Press the menu [Type 1].

(5) Sequence No. designation: Go to step (6).
Line No. designation: Press the menu [Specif.Line-No.].

(6) Set the position to restart the search in the input section. (*1) →

<Sequence No. designation>
Delimit the program No., sequence No., block No. and repeat count in the input section using "/".
<Line No. designation>
Delimit the program No., line No. and repeat count in the input section using "/".

<Note>
- When the [INPUT] key is pressed without entering data in the input area, restart search will be carried out for the last executed block.

(7) Press the [INPUT] key or the menu [Search exec]. → Restart search is executed. (*2)
The message "Executing restart search" appears during the search process, and the message "Restart search is completed" appears when completed.
Each axis' restart position and the restart remaining distance are displayed.
When the menu [MSTB history] is pressed, the MSTB history screen will open, and the M, S, T, B command used in the machining program will be listed.

(*1) The program to be searched for is different depending on whether or not the program No. is specified.
When the program No. is specified: The main or sub program is searched for.
When the program No. is not specified: The current program is searched for.

(*2) The maximum number of digits is eight for the sequence No. and nine for the block No.

Note

(1) When the sorting of tool data is in progress, the operation message "Sorting of tool data is in progress" appears, and the search cannot be executed.

10.7.7 Executing Restart Search (Restart Type 2)

If a machining program which differs from the machining program to be restarted was operated in the device NC memory, NC memory 2, HD, memory card, DS, or USB memory before starting restart search, the machining program will be restarted 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.
When using tape operation, index the top of the tape.

(4) Press the main menu [Restart]. → The sub-menu is displayed.
The main screen for restart search appears.

(5) Press the menu [Type 2].

(6) Press the sub-menu [Top search]. → The window for top search appears.

(7) Press the sub-menu [Device select].
Press the menu key for the device to execute a search.

(8) Use the [↑], [↓], PAGE, or PAGE key to align the cursor with the directory of target file, and press the [INPUT] key.

(9) Press the [↑], [↓], PAGE, or PAGE key to move the cursor to the program to be searched. → Set the cursor to 1000.

(10) Press the [INPUT] key or the menu [Search exec]. → The top search starts.
When the top search is completed, the operation message "Top search completed" appears.
The top search screen closes, and the main screen for restart search appears.

(11) Sequence No. designation: Go to step (12).
Line No. designation: Press the menu [Specif.Line-No.].

(12) Set the position to restart the search in the input section. (*1)

<Sequence No. designation>
Delimit the program No., sequence No., block No. and repeat count in the input section using "/". (*2)

<Line No. designation>
Delimit the program No., line No. and repeat count in the input section using "/". (*2)

(13) Press the [INPUT] key or the menu [Search exec].

Restart search is executed. (*3)

The message "Executing restart search" appears during the search process, and the message "Restart search is completed" appears when completed.
Each axis' restart position and the restart remaining distance are displayed.
When the menu [MSTB history] is pressed, the MSTB history screen will open, and the M, S, T, B command used in the machining program will be listed.

(*1) The program to be searched for is different depending on whether or not the program No. is specified.
When the program No. is specified: The main or sub program is searched for.
When the program No. is not specified: The current program is searched for.

(*2) If subprogram repeat count is designated to the block to be searched, set the count. 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.

(*3) The maximum number of digits is eight for the sequence No. and nine for the block No.

Note

(1) When the sorting of tool data is in progress, the operation message "Sorting of tool data is in progress" appears, and the search cannot be executed.

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.
When using tape operation, index the top of the tape.

(4) Press the main menu [Restart].

The sub-menu is displayed.
The main screen for restart search appears.

(5) Press the menu [Type 2].

(6) Press the sub-menu [File set].

The window for file set appears.

(7) Press the menu key for the device to execute a search.

(8) Use the [↑], [↓], PAGE, or PAGE key to align the cursor with the directory of target file, and press the [INPUT] key.

(9) Press the [↑], [↓], PAGE, or PAGE key to move the cursor to the program to be searched. → Set the cursor to 1000.

(10) Press the [INPUT] key or the menu [Search exec]. → The main screen for restart search appears.

(11) Sequence No. designation: Go to step (12).
Line No. designation: Press the menu [Specif.Line-No.].

(12) Set the position to restart search in the input section. (*1) →
<Sequence No. designation>
Delimit the program No., sequence No., block No. and repeat count in the input section using "/". (*2)
<Line No. designation>
Delimit the program No., line No. and repeat count in the input section using "/". (*2)

(13) Press the [INPUT] key or the menu [Search exec]. →
Restart search is executed.
The message "Executing restart search" appears during the search process, and the message "Restart search is completed" appears when completed.
Each axis' restart position and the restart remaining distance are displayed.
When the menu [MSTB history] is pressed, the MSTB history screen will open, and the M, S, T, B command used in the machining program will be listed.

(*1) The program to be searched for is different depending on whether or not the program No. is specified.
When the program No. is specified: The main or sub program is searched for.
When the program No. is not specified: The current program is searched for.

(*2) If subprogram repeat count is designated to the block to be searched, set the count. 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.

10.7.8 Executing Restart Search (Restart Search for All Part Systems in Batch)

When the multi-part system program management is enabled (#1285 ext21/bit0 = 1) and the search through a program for all part systems at once is enabled (#1285 ext21/bit1 = 1), the restart search is executed for all part systems with one ONBP number in one search operation.

(Example) When tool breakage during execution of O1000 N7 in the 2nd part system occurred and the restart search is executed from the O1000 N6 block in all the part systems

(1) Press the feed hold button in all the part systems and retreat to the tool change position by manual means or MDI.

(2) Press the reset key and suspend the present processing.

(3) Replace with a new tool.

(4) Press the main menu [Restart].

(5) Press the menu [Type 1].

(6) Set the position to restart search in the input section → Delimit the ONB number in the input section using "/".
<When O No. was attached>
The main program or sub program is targeted.
(Example) 1000/6/0
<When O No. was not attached>
(Example) /6/0

<Note>

- When the [INPUT] key is pressed without entering data in the input area, restart search will be carried out for the last executed block in each part system.

(7) Press the [INPUT] key or the menu [Search exec]. → Restart search is executed for all part systems in batch with the same ONBP number as that entered.

The message "Executing restart search" appears during the search process, and the message "Restart search is completed" appears when completed.

Each axis' restart position and the restart remaining distance of the displayed part system are displayed.

When the menu [MSTB history] is pressed, the MSTB history screen will open as a pop-up window, and M, S, T, B command of the the displayed part systems used in the machining program will be listed.

Note

(1) When the sorting of tool data is in progress, the operation message "Sorting of tool data is in progress" appears, and the search cannot be executed.

10.7.9 Returning to the Restart Position

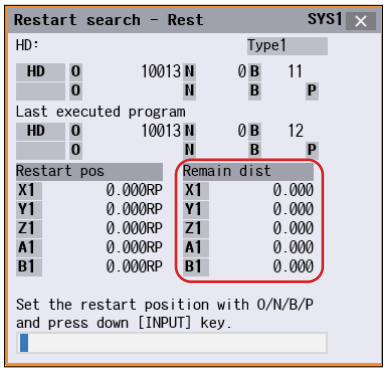
The axis is returned to the restart position after restart search is completed.
The method for returning to the restart position (manual/automatic) can be selected with the parameters "#1302 AutoRP".
0: Manual restart position return
1: Automatic restart position return

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".



(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.

(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.

(3) If even one axis has not completed return to the restart position at cycle start, the error "M01 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.

(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.

(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.

(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.

(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)
- When "#1100 Tmove" = "0"
  T command that selects a tool with different tool length/wear amount than the tool selected by the program.
- When "#1100 Tmove" = "1"
  A program error (P48) will not occur by T command.
- When "#1100 Tmove" = "2"
  T command that selects a tool with different wear amount than the tool selected by the program.

(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.

(2) Even if the parameter "#1302 AutoRP" is 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.

(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.

(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, an error occurs. The error "M01 Restart pos. return incomplete" is displayed.

(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 "M01 Restart pos. return incomplete" occurs.

(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.

(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)

- When "#1100 Tmove" = "0"
  T command that selects a tool with different tool length/wear amount than the tool selected by the program.
- When "#1100 Tmove" = "1"
  A program error (P48) will not occur by T command.
- When "#1100 Tmove" = "2"
  T command that selects a tool with different wear amount than the tool selected by the program

(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.

(9) When an axis moved to return is operated manually, the axis cannot return to correct position.

10.7.10 Executing the MSTB Commands

If the menu [MSTB history] is pressed after restart search is completed, the MSTB commands used for machining program appear. If the cursor is moved to the listed M, S, T or B command and the [INPUT] key is pressed, that command will be executed.
Up to 35 M commands, 3 commands each for S1 to S8, 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 sub-menu [MSTB history]. → The pop-up window changes to the MSTB history window. The MSTB commands used for machining program are listed.

(2) Using the [↑], [↓], [→] or [←] key, 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 grayed out. The cursor will move to the next item.

(4) Repeat steps (2) and (3).

(5) When completed with all settings, press the menu [Close] or [MSTB history]. → The MSTB history pop-up window closes and the restart search main window appears as a pop-up window.

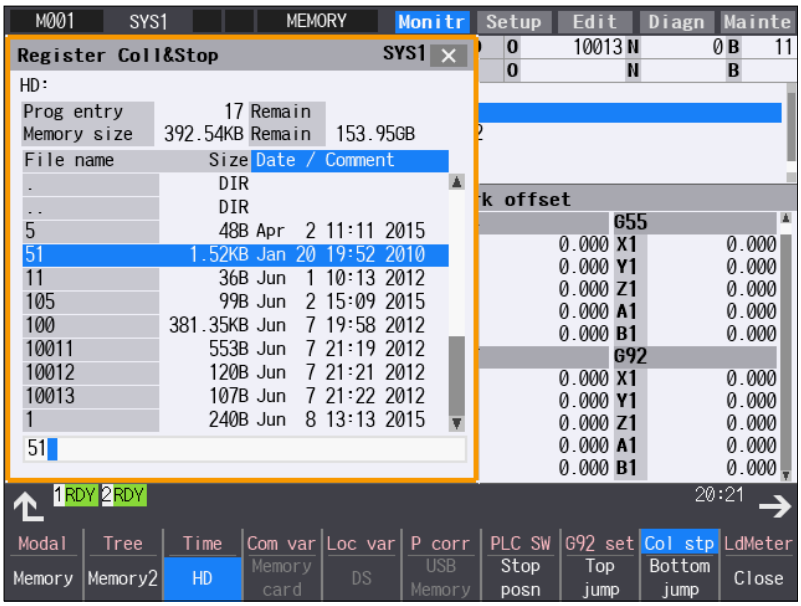
10.8 Collation and Stop

Monitr – Col stp

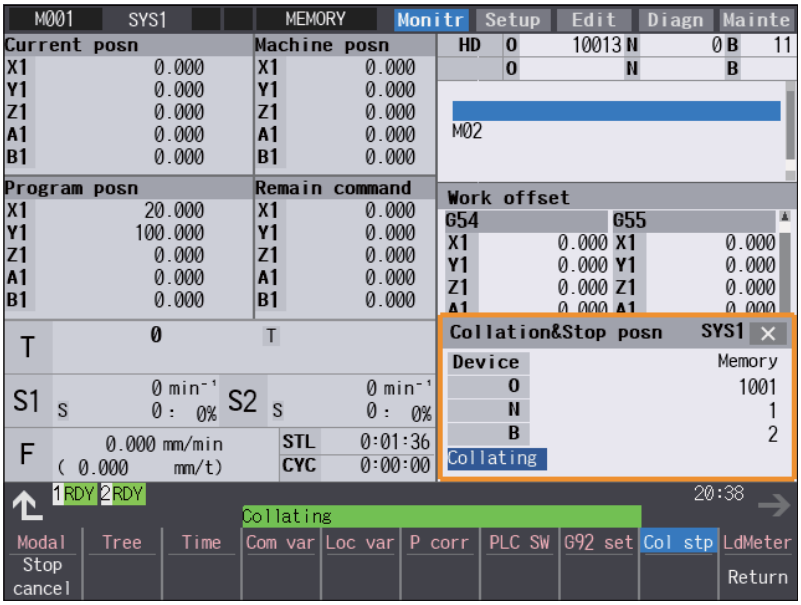
The machining program operation can be block stopped at a registered collation and stop position. The registered collation and stop position can be canceled.

When the specified block appears, NC status will change single block stop after the execution of the block.

■When collation and stop is registered



■When collation and stop position is displayed



Display items

Display items		Details
(1)	Collation and stop device name	This is the name of the device to be stopped for collation.
(2)	Collation and stop position	This is the position of the program where collation and 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 collation	This is displayed during collation.

Menus (When collation and stop is registered)

Menus	Details
Memory	This selects the device to be stopped for collation. When a device with directory is selected, the root is selected first. This menu [HD] is not displayed on M800VS/M80V Series. (*1)
Memory2	
HD	
Serial	
Memory card	
DS	
USB Memory	
Stop posn	This opens the collation and stop position display window as a pop-up window.
Top jump	Display starts from the first page of the directory/list of files. The cursor moves to the top line.
Bottom jump	The last page of the directory/list of files is displayed. The cursor moves to the bottom line.
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.
Preview	This switches to show or hide the preview (detailed information).
Change Disp	This changes the display format of the file/directory list.
List update	This updates the list contents. (The latest contents of the currently selected device and directory are listed.)
Sort change	This changes the method of sorting the list.
Close	This closes the pop-up window and quits this function.

(*1) When the HD mode operation on an industrial computer (IPC) is enabled with M80V Series, the menu [HD] becomes selectable and you can select files in the hard disk on the industrial computer. The availability of this function depends on the MTB specifications (parameter "#1760 cfgPR10/bit2").
Set the IP address of the industrial computer to the parameter #11005 to use this function.

Menus (When collation and stop position is displayed)

Menus	Details
Stop cancel	This cancels the collation and stop setting. Note that automatic operation is not reset.
Return	This returns the list when collation and stop is registered.

Operation method (Registering the collation and stop)

(1) Press the $\leftrightarrow$ key to select the part system to be stopped for collation. → The currently selected part system is displayed at the upper left of the screen.

(2) Press the main menu [Col stp]. → The mode changes to the buffer correction mode.

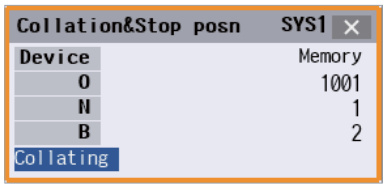
(3) Select the device.
(Example) [Memory] → The selected device name and root directory (memory:/) are displayed in the device name and directory display fields.
The menu [Memory] is highlighted.

(4) Press the [↑], [↓], PAGE, or PAGE key to move the cursor to the directory containing the file to be set.

If the list contents differ from the actual device or directory, press the menu [List update].

(5) Press the [INPUT] key. → Moves into the directory.

(6) Input the program No., sequence No. and block No. using / as a delimiter.
(Example) 1001/1/2 [INPUT]
When carrying out collation and stop of MDI program, set "0" as the program No.
If pointing the cursor to a file name, the file name is echoed back to the input area. → The collation and 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 collation and stop is completed, a message indicating the end appears.
The stop position displayed on the screen is cleared, and the "Verifying" display is erased.

Press the menu [Stop cancel] to cancel the collation and stop.

Note

(1) The maximum number of digits is eight for the sequence No. and nine for the block No.
(2) Collation and stop can be canceled during automatic operation.

Precautions

(1) When registering the stop position (O, N, B values), confirm that the block exists in the searched program. Collation and 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) When the parameter "#1754 cfgPR04/bit6" is "1", collation and stop is not performed for a block with N No. command only.

(6) If there are several sequences and blocks with the same number in one program, collation and stop will be executed after executing the first block that matches in the execution order.

(7) The settings are canceled when collation and stop is executed.

(8) The settings are canceled when reset is executed.

(9) If only the program number is set, collation and stop will take place at the head of the program only when there is a program number at the first line.

(10) Collation and stop cannot be performed or canceled for blocks being executed or blocks already read into the preread buffer.

(11) Collation and stop is not performed in the tapping mode.

(12) Collation and stop is possible in a subprogram, but is not possible in a MTB macro program.

(13) If collation and stop is set for a fixed cycle block, it will be executed after the positioning block is completed.

(14) Collation and stop is possible even when editing is locked.

(15) Collation and stop is executed after executing the block set for the collation and stop position. If the position is set in a program call (M98) block, collation and stop will be executed before the subprogram is called.

(16) If the set block is set to be skipped, collation and stop will not be executed.

(17) The collation and stop position can be registered for each part system.

(18) The stop position cannot be registered during verification.

(19) Registration of collation and stop position is not possible for a program with 33 or more file name characters.

(20) In the case where the device is HD, memory card, DS, or USB memory at the time of collation and stop position registration, inputting "0//" will be resulted in a setting error.

(21) When the collation and stop position has been set in the following conditions, the collation and stop is performed for the operation searched program.
(If the operation search target device is the same as the collation and stop target, the collation and stop is performed with the contents correctly set.)

(Example) When the same name programs exist in the NC memory and HD

(a) Operation searched program -> NC memory: 100.PRG
Collation and stop position set program -> HD: 100.PRG

(b) Operation searched program -> HD: 100.PRG
Collation and stop position set program -> NC memory: 100.PRG

(The collation and stop is performed for the NC memory in the condition (a) and performed for the HD in the condition (b).)

(22) Collation and stop position for sub part system I/II cannot be registered for M80V Series. If it is registered, the message "Search error" appears.

(23) Collation and stop position for sub part system II cannot be registered for M800V Series. If it is registered, the message "Search error" appears.

10.9 Correcting the Machining Program

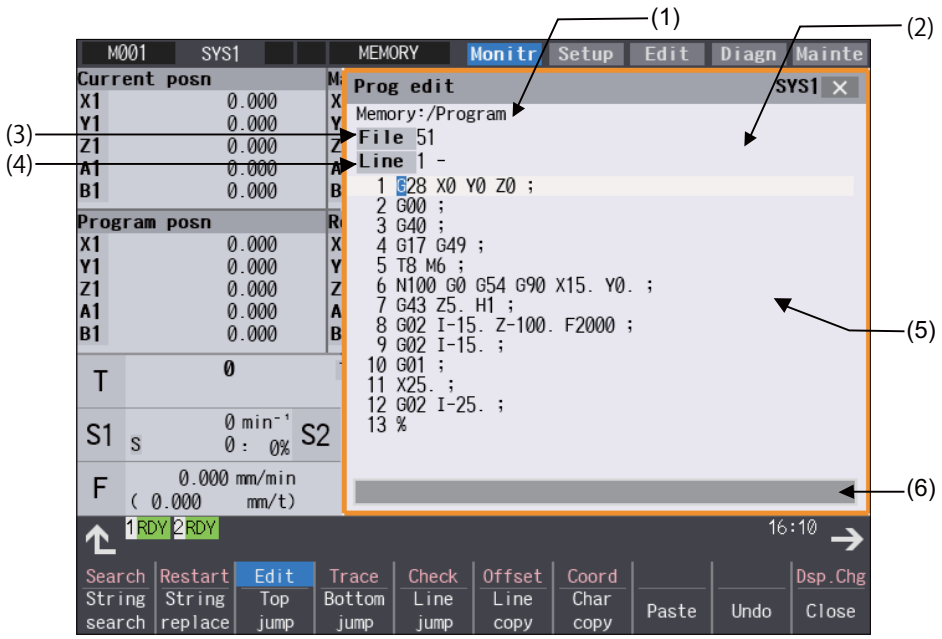
Monitr – Edit

10.9.1 Program Editing

If you press the [Edit] main menu or set the edit/search signal ON, the program editing window appears to enable you to edit a program. However, if the target program is not found (displays "O0") or programs are serially searched, the editing window does not appear. The program is opened on the unselected side for the 2-part system simultaneous display.

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.
When the [INPUT] key is pressed, the program is saved in the NC memory and the "Editing" message disappears.

A program to be edited can be switched to the other part system's program with $\leftrightarrow$ key when a parameter "#8952 Edit-win $ switch" is set to "1".
For information on editing operations, refer to "3.2 Creating and Editing a Machining Program".



Display items

	Display items	Details
(1)	Path display	The path of the currently opened program file is displayed. (If the path is too long, the portion exceeding single-byte 49 characters is not displayed.) (Example) Memory: /Program, HD: /PRG/SUB
(2)	Edit type display	For mass-editing, "EX" appears. For regular editing, nothing appears.
	Current edit display "Editing"	Displayed when the program is edited after it was displayed.
	Insert mode display	The overwrite mode and insert mode are switched by pressing the [INSERT] key. During insert mode: "INS" appears. During overwrite mode: Nothing appears.
(3)	File name display	This displays the file name of the program currently being edited. "MDI" appears when the MDI program is edited.
(4)	Regular editing: top line of the displayed program	The top line of the program to be displayed is displayed.
	Mass-editing: display position	During mass-editing, the position of the currently displayed page is displayed in percentage (rate) when the entire size of the program is assumed to be 100%.
(5)	Line number	Each line of the program is numbered, and only the last three digits are displayed. If a line is wrapped because a line is too long, no number is assigned to the top of the next line. For mass-editing, line number does not appear.
	Program display	This displays the contents of the program (machining program, MDI program) currently being edited. The line that the cursor is on is highlighted. (During mass-editing, the line is not highlighted.) If the value of the parameter "#8952 Edit-win $ switch" is set to "1" and a part system is switched while the window is displayed, the display is changed to the program of the part system. Machining program editing: Switch the mode selection switch to other than "MDI mode" and press the [Edit] main menu. However, to edit data, designate the machining program to execute a search. MDI program editing: Switch the mode selection switch to "MDI mode" and press the main menu [Edit].
(6)	Input section	This displays details of the key input.

A selection part system operation search can be carried out using the block with the cursor as the top.
There are 2 ways to search cursor position, "INPUT search" and "Edit/search signal search".

INPUT search

By pressing [INPUT] key when it is in the normal status (not being in status such as editing or character searching, etc.), operation search of the selected part system is carried out starting from the block that has the cursor.

This is enabled when "1" is set to the parameter "#11031 Cursor pos search".

(1) Carry out the operation search to find the program to carry out the cursor position search.

(2) Press the menu [Edit]. → The program edit window appears. The program is displayed starting from the block currently searched.
(During automatic operation or checking, the program is displayed from its top.)

(3) Move the cursor to the block to be carried out the cursor operation search.

(4) Press the [INPUT] key. → Operation search is carried out starting from the block that the cursor is positioned. During the search "Searching" will display blinking. When the search is completed, "Search completed" is displayed.

Note

(1) When operating character string search, string replace, cursor on input unit, or editing it will not start cursor position search by pressing [INPUT] key.

(2) A selection part system operation search can be carried out using the block with the cursor as the top.

(3) Operating search program will be the program that was searched when program edit window was open.

(4) Cursor position search is disabled during automatic operation, checking or restart search. However, it is enabled when the parameter "#11039 Cusr pos srch type" is 1 even if the operation is under single block stop. For details, refer to "Search during single block stop" in "10.9.1 Program Editing".

(5) When MDI program is displayed, the block with the cursor will be set for MDI by pressing [INPUT] key.

(6) If the [INPUT] key is pressed when the data protection is valid, the cursor position search will be executed without displaying the message "Data protect", and the message "Search completed" will appear upon completion.

(7) If the INPUT key is pressed during automatic operation, check, or restart search while the program currently being displayed is not in a selected part system, message "Executing automatic operation", "Program checking" or "Program restarting" is not displayed. The cursor position search is carried out, and message "Search completed" is displayed.

(8) If the N number of the block to be searched exceeds the maximum value or is set to "0", it causes a search error in all blocks that belong to the N number.

(9) If characters other than digits follow "N" of the block to be searched, "N**" of the program is handled as a normal block, not a sequence number. Therefore, if the target block is searched, it is handled as a block that belongs to the previously searched N number.

Edit/search signal search

This is enabled when "2" or "3" is set to the parameter "#11031 Cursor pos search".

(1) Carry out the operation search to find the program to carry out the cursor position search.

(2) Turn ON the edit/search signal. → The "Edit/Search" window is displayed. The program is displayed starting from the block currently searched.
(During automatic operation or foreground checking, the program is displayed from its top.)

(3) Move the cursor to the block to be carried out the cursor operation search.

(4) Turn OFF the edit/search signal. → The "Edit/Search" window is closed, and the operation search is carried out starting from the block that has the cursor.
When the search is completed, "Search completed" is displayed.
(Also in the case where the screen is changed with a function key, etc. and the "Edit/Search" window is closed, the operation search is carried out.)

Note

(1) If the "edit/search signal" is set to ON on a screen other than the monitor screen, the "Edit/Search" window is not displayed. To display the "Edit/Search" window, switch to the monitor screen. Switching to the monitor screen automatically displays the "Edit/Search" window.

(2) If a window (operation search window, menu list, guidance, custom window, etc.) other than the program edit window is displayed while the monitor screen is being displayed, the "Edit/Search" window is not displayed even if the edit/search signal is set to ON. To display the "Edit/Search" window, close the currently displayed window. When the currently displayed window is closed, the "Edit/Search" window is displayed automatically.

(3) If the edit/search signal is set ON during display of the program edit window on the monitor screen, the display changes to the "Edit/Search" window while the editing contents, cursor position, and menu etc. remain unchanged.

(4) If the "Edit/Search" window is attempted to be closed during editing, a message to confirm whether to save the data will appear.
When it appears, if save is carried out by pressing the [Y] or [INPUT] key, the operation search will be carried out starting from the block that has the cursor.
If save is not carried out, the edited data will be aborted, and the operation search will not be carried out.

(5) If the "Edit/Search" window is closed while the cursor is positioned in the input area during string searching or string replacement, the operation is canceled, and the cursor returns to the editing area. After this, an operation search is carried out with the block located by the cursor at the top. However, during editing, the cursor returns to the editing area, and the same operation as that described in (3) is performed.

(6) If the "Edit/Search" window is displayed while the automatic operation, check, or restart search is in process in the selected part system, message "Executing automatic operation", "Program checking" or "Program restarting" is displayed. However, when "#11039 Cusr pos srch type" is set to "1", cursor position search is possible even in the single block stop state. For details, refer to "Search during single block stop" in "10.9.1 Program Editing".

(7) If the "Edit/Search" window is closed during automatic operation, program check, or restart search in the selected part system, message "Executing automatic operation", "Program checking" or "Program restarting" is displayed, but an operation search is not carried out. However, when "#11039 Cusr pos srch type" is set to "1", cursor position search is possible even in the single block stop state. For details, refer to "Search during single block stop" in "10.9.1 Program Editing".

(8) If the "Edit/Search" window is opened while the operation mode is set to the MDI mode, the MDI program is displayed in the "Edit/Search" window. In this state, if the "Edit/Search" window is closed, the MDI setting at the cursor position is executed.

(9) The part system that is selected when the "Edit/Search" window is opened is to be searched when the "Edit/Search" window is closed.

(10) The program that is searched when the "Edit/Search" window is opened is to be searched when the "Edit/Search" window is closed.

(11) In the following cases, edit/search is not available with the "Edit/Search" window. After an error is displayed, a blank "Edit/Search" window is displayed.

- Edit lock B, C is enabled
- Operation search is not carried out
- Edit file size exceeded
- A serial program is being searched

(12) If the edit/search signal is set ON at power ON, the "Edit/Search" window is displayed.

(13) The "Edit/Search" window cannot be closed with the cancel key. To open another window of the monitor screen, either turn OFF the edit/search signal or use the direct screen selection in the menu list.

(14) If you switch to the monitor screen to edit using the direct screen selection while the "Edit/Search" window is displayed, the "Edit/Search" window is closed once, a search is carried out, and then the "Edit/Search" window is displayed again.

(15) If the N number of the block to be searched exceeds the maximum value or is set to "0", it causes a search error in all blocks that belong to the N number.

(16) If characters other than digits follow "N" of the block to be searched, "N**" of the program is handled as a normal block, not a sequence number. Therefore, if the target block is searched, it is handled as a block that belongs to the previously searched N number.

Search during single block stop

Cursor position search can be operated during single block stop when "1" is set to the parameter "#11039 Cusr pos srch type". By pressing menu [Edit] during single block stop, single block stop position becomes top of the program display.

(1) Pause automatic operation with single block stop.

(2) Press the menu [Edit]. → The program edit window appears.
The block where the operation is stopped due to single block stop is displayed on the top.
However, for mass-editing, when the program that can be displayed within one page on the edit screen, or when the program has been in single block stop status at the block existing in the first or last page, the cursor is displayed at the stopped block. In these cases, the top or end of the program is displayed, not the stopped block at the head.
If the operation has been stopped in a subprogram, that subprogram is displayed.

(3) Move the cursor to the block to be carried out the cursor operation search.

(4) Press the [INPUT] key. → The search will operate by single block stop position with top.
During the search "Searing" will display blinking.
When the search is completed, "Search completed" is displayed.

Note

(1) Blocks lower than the point of the single block stop only are eligible for cursor position search designation during single block stop. "Search error" will display and the search will fail when cursor position search is attempted at an already executed block.

(2) The blocks between single block stop position and search position block will be ignored when cycle start is operated after cursor position search is operated during single block stop.

(3) While the part system with a cursor position search to be performed is in block stop state in arbitrary reverse run mode (including a case in which the reverse run control mode signal is set ON), message "search error" is displayed, and a cursor position search cannot be carried out.

(4) If the part system with a cursor position search to be performed is placed in block stop state due to a program error, message "During automatic operation" is displayed, and an operation search is not carried out.

(5) If cursor position search is commanded at a block that causes a program error, the block of the single block stop, instead of the cursor position search target block, will appear on the top of program buffer display upon automatic start up.

(6) If cursor position search is commanded at a block that comes later than a fixed cycle while stopped with single block stop in the midst of the fixed cycle, the cursor position search target block will appear on the top of the program buffer display. However, automatic operation will start from the top of the remaining of the fixed program, and after finishing the fixed cycle, will move on to executing the cursor search target program.

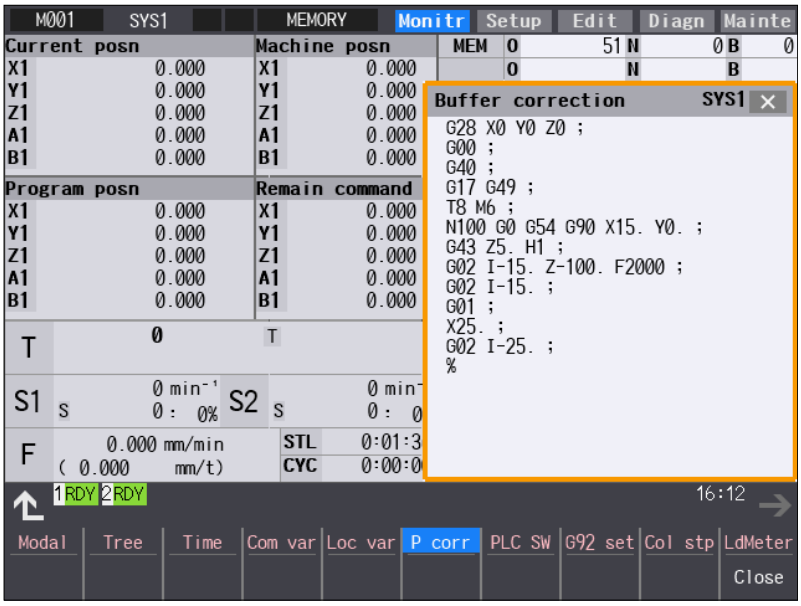
(7) If a cursor position search is carried out for blocks after the MTB macro program while the MTB macro program is interrupted by a single block stop, a program buffer is displayed from a search block. However, when the system is started automatically, the MTB macro program is executed continuously, and the block detected by a cursor position search is processed after the MTB macro program has been completed.

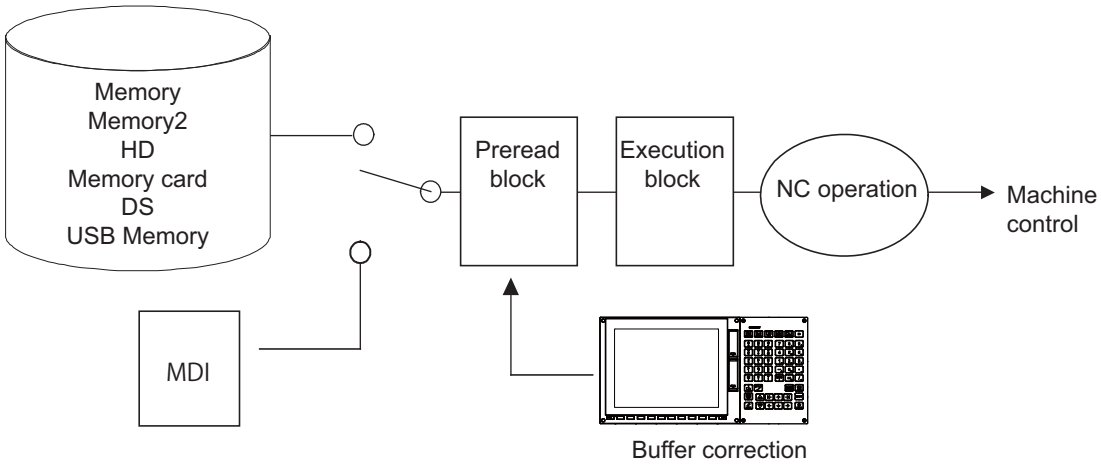
10.9.2 Buffer Correction

Monitr – P corr

During automatic operation (NC memory, NC memory 2, HD, DS, memory card or USB memory) 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.





(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.)

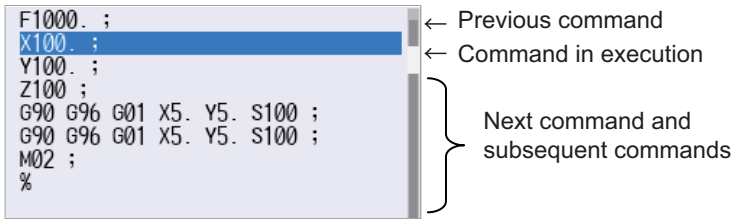
(3) Several blocks following the next command can be corrected simultaneously.

Menus

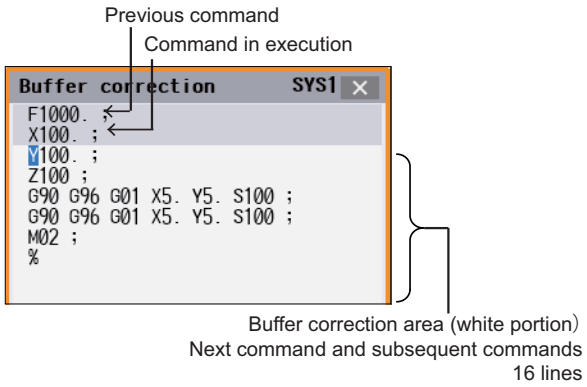
Menus	Details
Close	This closes the pop-up window and quits this function.

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.



(1) Press the main menu [Prg correct]. → The mode changes to the buffer correction mode.



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 [INPUT] 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

■Buffer correction mode will not be applied in the following cases:

(1) Machining program for buffer corrections is in edit lock or program display lock state.

(2) When the data protection is valid.

(3) Machining program for the device in write-protection state

(4) Machining program in read-only state

(5) During the compound type fixed cycle, the tool center point control or executing the MTB macro program

■Starting and ending buffer corrections

(1) 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.

(2) To stop or to cancel the buffer corrections, press the menu [Close] again, or press the  or  key.

(3) If another menu key, etc., is pressed before pressing the [INPUT] key, the corrected buffer contents return to the original data.

(4) If changing to another screen or resetting the NC during buffer correction or changing system, the buffer correction is canceled. In this case, the corrected contents are not saved, and the buffer correction mode ends.

■Display data during buffer correction

(1) 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. For the 15-type/19-type display unit, 28 blocks are displayed.

(2) Number of characters in one block is max. 256 (including ";" (EOB)). If exceeds 256 characters, the block will be split into two.

(3) Up to 4096 characters can be edited with buffer corrections.

■Operations during buffer correction

(1) If there is no ";" (EOB) at the data in the last block corrected when the [INPUT] key is pressed, it is added automatically.

(2) When the buffer correction mode is set, 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 [C.B] key, the block is deleted up to the section following the incomplete display (up to the ";").

(3) Operation cannot be started during buffer correction. An alarm occurs.

(4) 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.

(5) Buffer correction writing is not interrupted by NC reset.

(6) The operation 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.

■For multi-part system

(1) 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.

(2) Note that if the same program is executed in another part system after a buffer correction, there may be some changes in paths, etc.

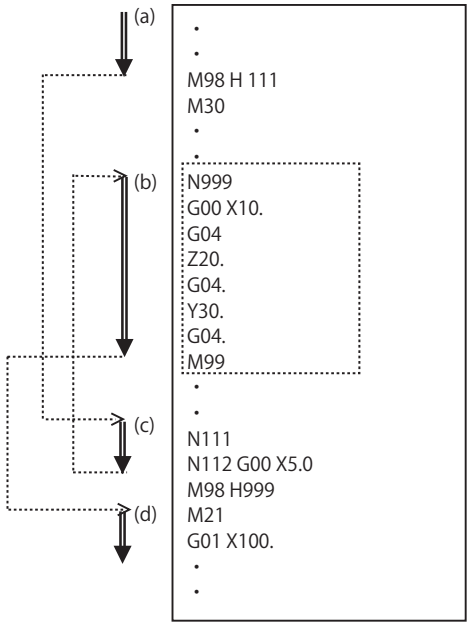
(3) The program which has searched for the sequence No. while operating in another part system cannot be buffer correction. The operation message "Buffer correct not possible" appears.

■**Previous command and command in execution**

(1) 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.

(2) 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.

(3) The buffer cannot be corrected when the block where the buffer is to be corrected is in the forward of the block called at another nest level. The operation message "Buffer correct not possible" appears.
The buffer can be corrected when the block where the buffer is to be corrected is in the backward of the block called at another nest level.



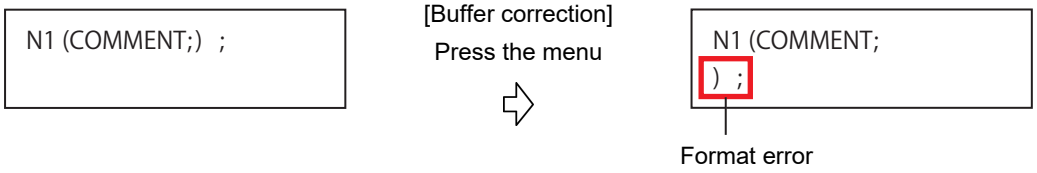
(a): The buffer can be corrected.

(b), (d): The buffer can be corrected because the block is in the backward of (a).

(c): The buffer cannot be corrected because the block is in the forward of (b).

(4) With buffer correction, as EOB (;) in the comment block is treated as a line feed code, when edited and saved with buffer correction, the program error occurs.

(Example)

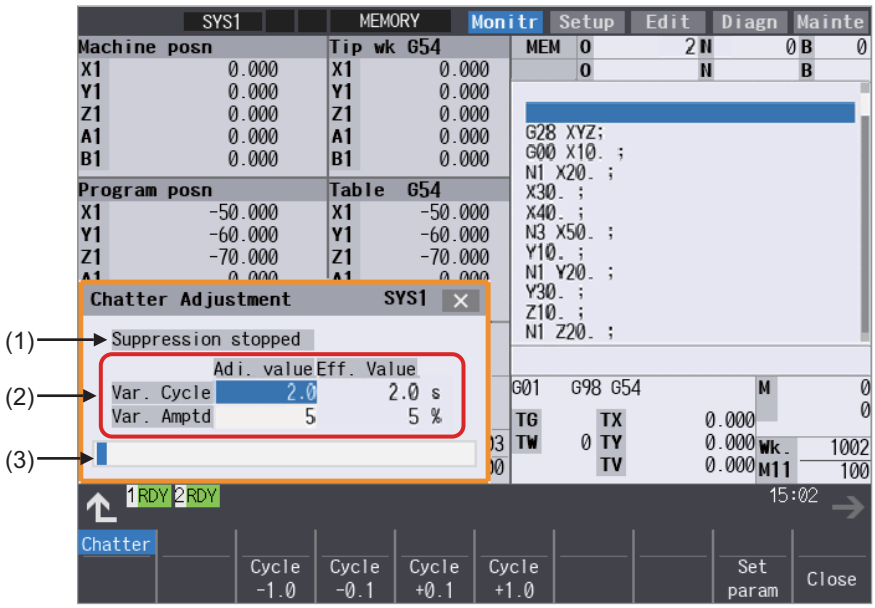


(5) When the parameter "#19006 EOR Disable" is set to "1", the contents which follows "%" in the program cannot be corrected even though they are displayed.

10.10 Chatter Suppression

Monitr – Chatter

This screen displays the status of chatter suppression. You can adjust the parameters to suppress the chatter.



Display items

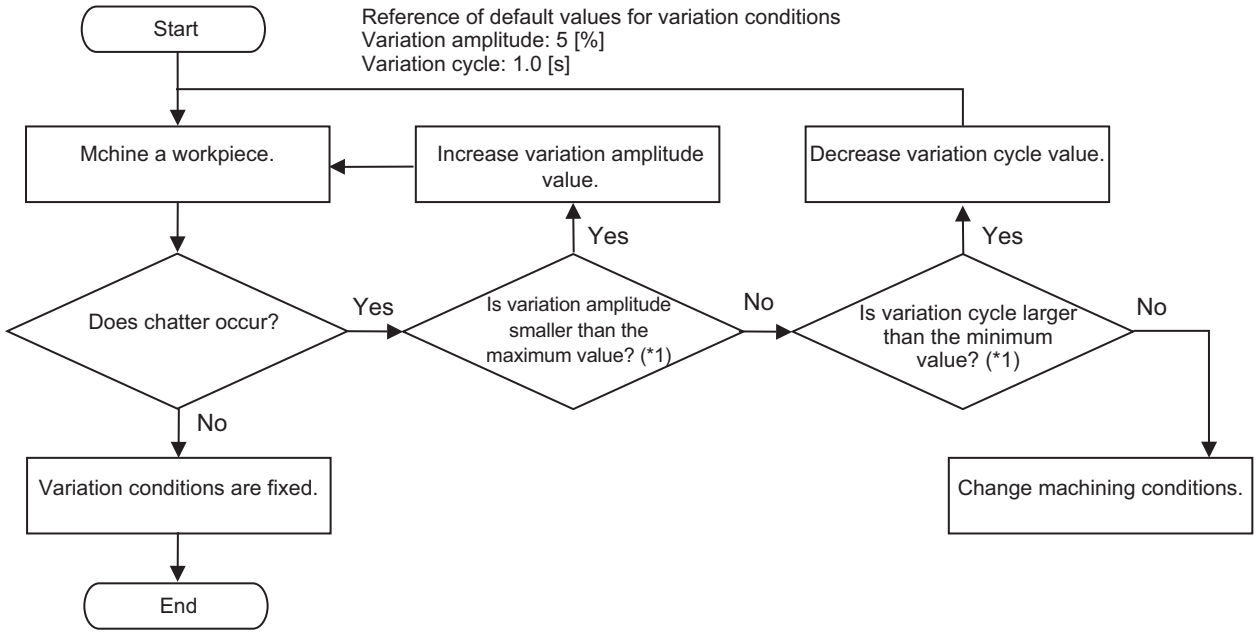
	Display items	Details
(1)	Chatter suppression status	Displays the chatter suppression status
(2)	Variation conditions	Displays the adjustment value and effective value of the variation condition. Input range of adjustment value is as follows. Note that the chatter is not suppressed if you set "0" for the variation cycle or variation amplitude. Variation cycle: 0.0 to 5.0 [s] Variation amplitude: 0 to 50 [%]
(3)	Input section	Displays the input contents when the variation conditions are set.

Menus

Menus	Details
Cycle -1.0 Amptd. -10 Cycle -0.1 Amptd. -1 Cycle +0.1 Amptd. +1 Cycle +1.0 Amptd. +10	Changes the setting value of the variation cycle or variation amplitude by the amount displayed on each menu. Depending on the cursor position, menus corresponding to the variation cycle or variation amplitude are displayed.
Set param	Reflects the setting values of the variation cycle and variation amplitude to the variation condition parameters.
Close	This closes the pop-up window and quits this function.

10.10.1 Adjusting the Variation Conditions

The chart below indicates the adjustment procedure when the chatter occurs.



(*1) The maximum value of the variation amplitude and the minimum value of the variation cycle are different from the parameter setting range.
The maximum value of the variation amplitude is difference between the clamp speed and the command speed.
In addition, if scratches occur, check the condition of the workpiece.

Operation method

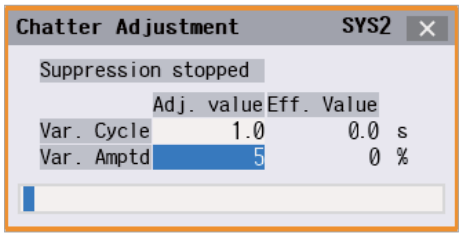
(Example) When chatter occurs in $2, adjust the parameters to suppress the chatter.

(1) Press the menu [Chatter] on the monitor screen.

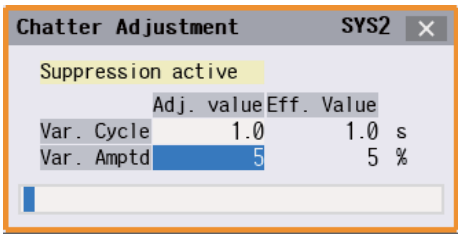
(2) Press the [$↔$] key to switch to the part system where the chatter occurs. Switch to $2 in this case.

(3) Input the adjustment values to the variation cycle and variation amplitude. → The adjustment values of variation conditions are set.

Reference of default values for variation conditions
Variation cycle: 1.0 [s]
Variation amplitude: 5 [%]



(4) Press the cycle start button to proceed the program to the block where the chatter occurs. → Actual variation conditions are displayed in effective values.



(5) Adjust the variation conditions in the block where the chatter occurs by pressing the menus.

(6) Press the menu [Set param]. → The operation message "Set up parameter?(Y/N)" appears.

(7) Press the [Y] or [INPUT] key.
Press the [N] key not to set the parameters. → The adjustment values are reflected to the following parameters.

- #19446 Chatter Supp Ampl
- #19447 Chatter Supp Cycl

Note

(1) Check the effective values, not the adjustment values, for actual variation conditions. The adjustment value and the effective value may not match due to the following factors.

- It took time that the adjustment values were reflected in the effective values after they were changed.
- The effective value of the variation amplitude was clamped.

(2) Changes for variation conditions made on the chatter adjustment screen are overwritten with other values during the following operations.

- If you change the variation conditions from system variables, the adjustment values on the chatter adjustment screen are overwritten with that of the system variables.
- During automatic operation, the adjustment values on the chatter adjustment screen are overwritten with the value of the variation condition parameters.
- When the setting values of the variation condition parameters are changed, the adjustment values on the chatter adjustment screen are overwritten with the changed values of the parameters.
- When the power is turned OFF and ON, the adjustment values on the chatter adjustment screen are overwritten with the values of the variation condition parameters.

(3) If the speed after the variation exceeds the clamp speed, the CNC automatically reduces the variation amplitude not to exceed the clamp speed.

11

Troubleshooting

This chapter explains remedies when any alarm message is displayed, and how to check the NC status.

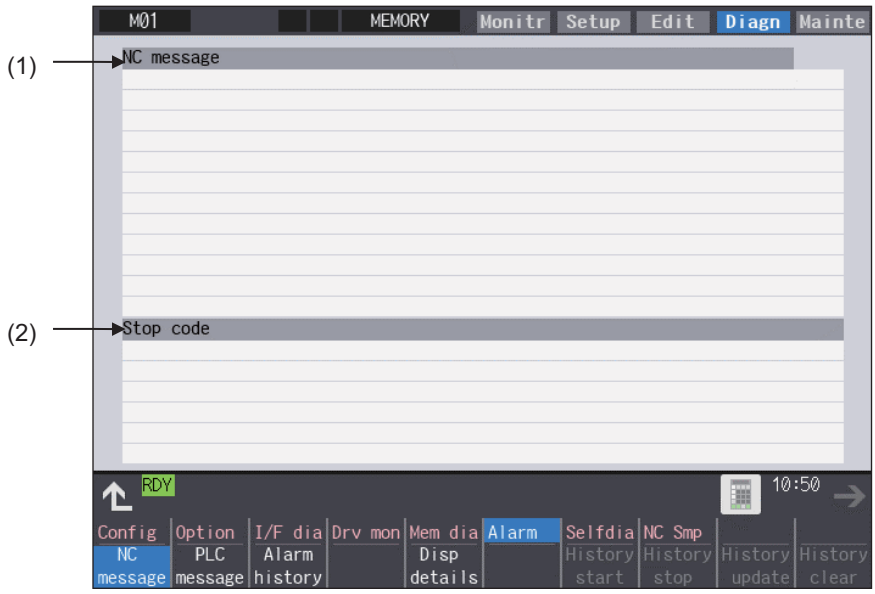
The operations are mainly performed on the diagnosis (Diagn) or maintenance (Mainte) screen.

11.1 Confirming the NC Message (Alarm Screen)

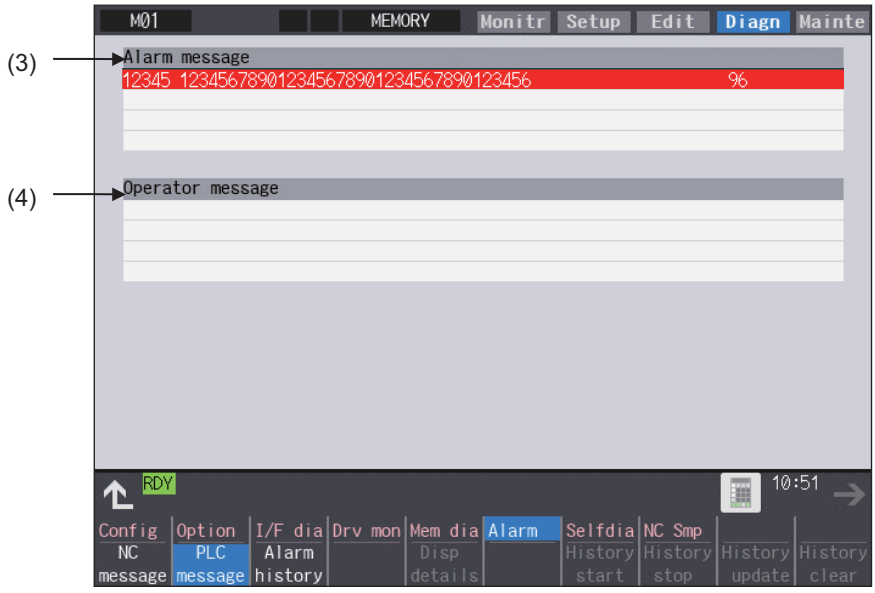
Diagn – Alarm

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 items		Details
(1)	NC message	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. This varies depending on the MTB specifications. Up to 4 messages are displayed.
(4)	Operator message	Using the PLC programs, this displays operator information messages. This varies depending on the MTB specifications. Macro alarm messages are also displayed in this field. Up to 4 messages are displayed.

Note

(1) When the 2nd line of the operator message cannot be displayed, only the 1st line is 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
NC alarm message at the background check		White	Orange

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.

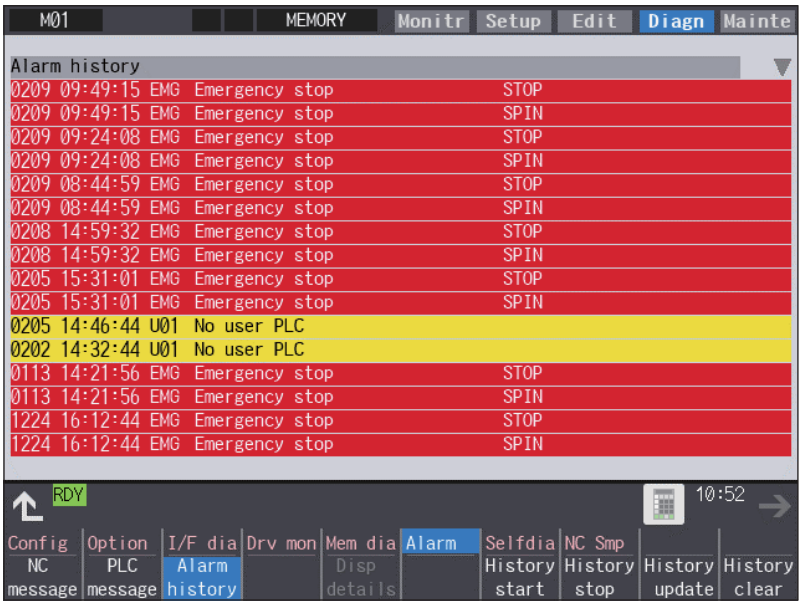
Axis type	Axis name display	Display example	Remarks
NC axis	Control axis name (Name of axis in part system)	XYZ	If the same message occurs for each part system, several NC axes are displayed together.
Spindle	'S' + spindle No.	S1S2	If the same message occurs, several spindles are displayed together.
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.

11.1.1 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 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), PLC alarm message

Note

(1) The following alarms are not recorded on the alarm/warning screen.

♦Operation alarm

- External interlock axis exists (M01 0004)
- Cutting override zero (M01 0102)
- External feed rate zero (M01 0103)
- Block start interlock (M01 0109)
- Cutting block start interlock (M01 0110)
- Interference check disabled (M01 0200)
- Rapid override zero (M01 0125)
- Sp-Sp polygon cut interlock (M01 1033)

♦"U50 PLC stopped" before the HMI screen starts up

(2) When multiple servo alarms have occurred, only an alarm which occurred first is displayed on the alarm/warning screen.

Menus

Menus	Details
Alarm history	This displays the first page of the alarm history. The history sequentially displays 16 alarms (20 alarms for 15-type/19-type display unit) per page with the latest alarm on the top. When the total number of alarms exceeds the maximum number, the second page will be displayed. If there are two or more NC alarms of the same day and time, the alarms are sequentially displayed from the alarm and warning of the 1st part system. To display the old history, press the PAGE key. To display the new history, press the PAGE key. The screen can also be switched by pressing ▲/▼ keys on it.
History start	The data collection of the alarm history is started (restarted). The operation message "The collection begin?(Y/N)" appears. When the [Y] or [INPUT] key is pressed, the data collection is started after the operation message "The collection begin" appears. Press the [N] or other than [INPUT] key when the data collection will be not started/restarted.
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] key is pressed, the data collection is stopped after the operation message " The collection stop" appears. Press the [N] or other than [INPUT] key when the data collection will be not stopped.
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.
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] 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.

Note

(1) The menus [History start], [History stop], [History update] and [History clear] are valid when the alarm history function is valid and the menu [Alarm history] is selected.

(2) When the alarm history function is used for the first time, clear the alarm history contents by pressing the menu [History clear]. Unnecessary data may be recorded in the alarm history.

Precautions

(1) If the recording of the alarms is stopped with the menu [History stop] in the alarm history screen, alarms are not recorded in the history even after the NC power is turned OFF and ON. In this case, the recording of the alarms must be started with the menu [History start] in the alarm history screen.

(2) If the time and date are changed, the change is not reflected to the time and date of the history data recorded before the change. Therefore, if the time and date are changed, clear the history with the menu [History clear].

(3) If an alarm occurs while the alarm history is displayed, the display is not updated. Update the history by switching the screen with the menu [History update].

(4) Depending on the part system setting of "Sampling spec" in the maintenance diagnostic data collection setting, the part systems to be recorded in the alarm history vary. If "0" is set, alarms of all part systems are recorded.

(5) The alarm history data collection start/stop operation is switched using the menu [Start] or [Stop] of data collection regardless of whether "Alarm history" is set to "0" or "1" in "Collecting data select" in the maintenance diagnostic data collection setting.

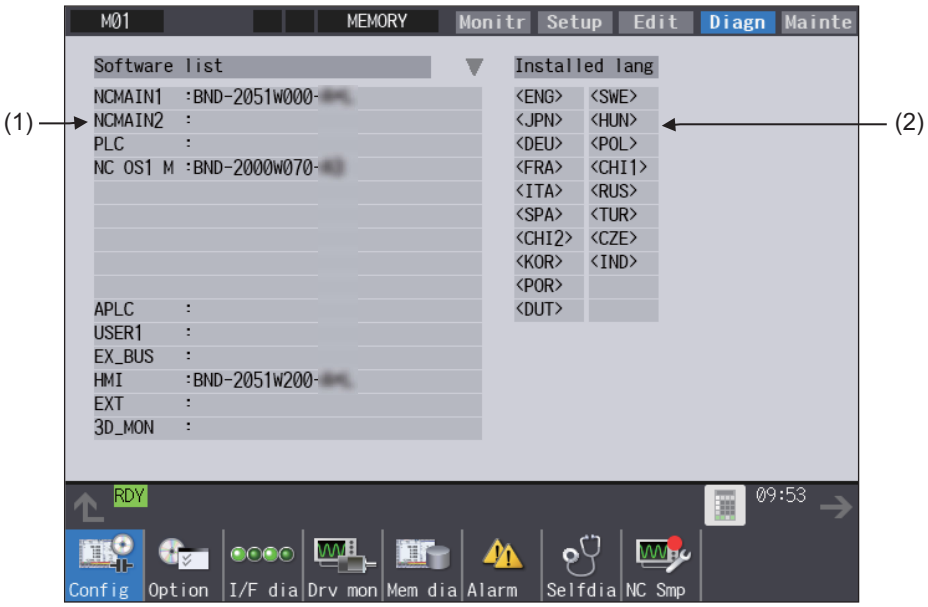
11.2 Checking Machine Status

11.2.1 System Configuration Screen

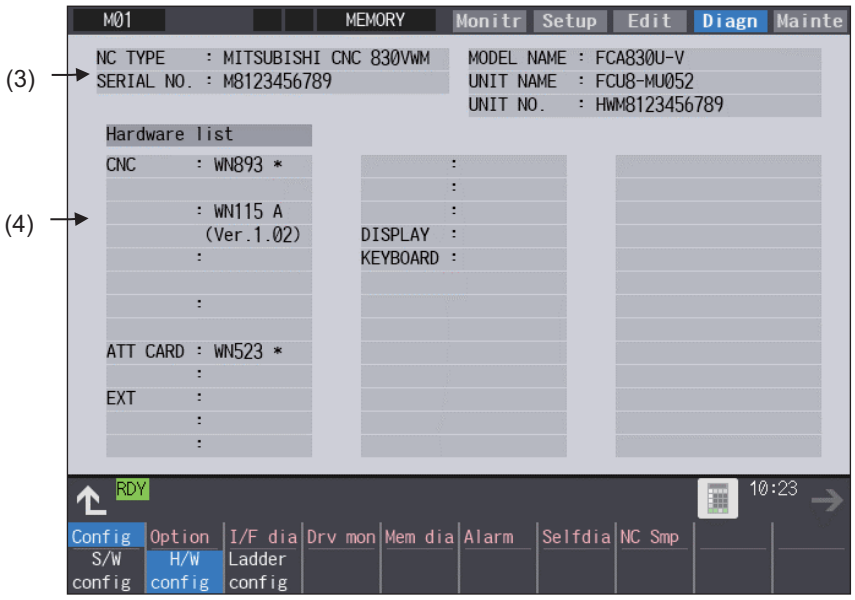
Diagn – Config

Select the menu [Config] on the diagnosis (Diagn) screen to display 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).

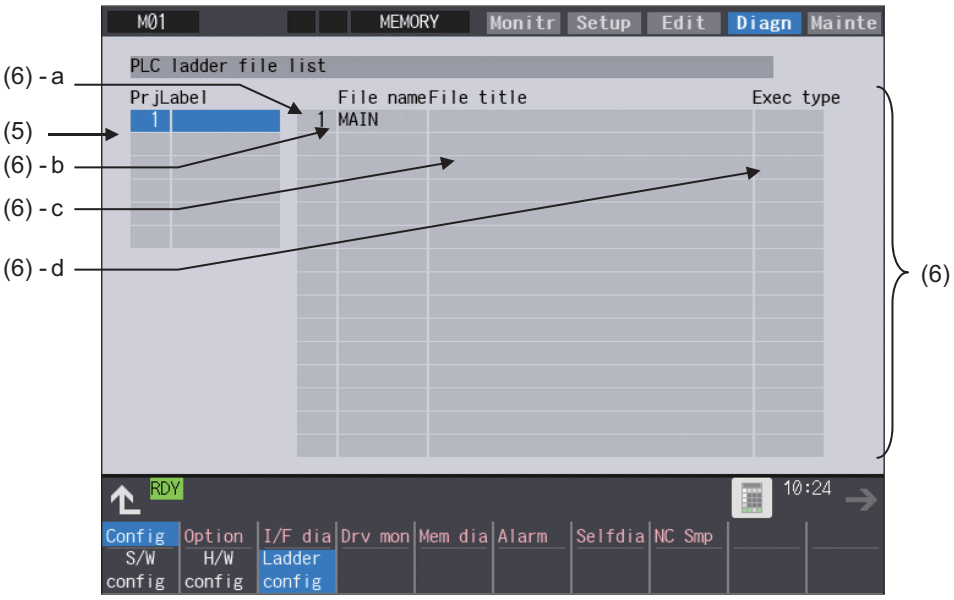
■Software configuration



■Hardware configuration



■PLC program configuration



Display items

Display items	Details		Remarks
(1) Software list	This displays a list of the software being used. See the next/previous page by using PAGE/PAGE keys, or ▲/▼ keys on the screen.		▲/▼ keys appear only when the screen has items on the next/previous page.
(2) Installed lang	This displays a list of installed languages. <ENG>: English <JPN>: Japanese <DEU>: German <FRA>: French <ITA>: Italian <SPA>: Spanish <CHI2>: Traditional Chinese <KOR>: Korean <POR>: Portuguese <DUT>: Dutch	<SWE>: Swedish <HUN>: Hungarian <POL>: Polish <CHI1>: Simplified Chinese <RUS>: Russian <TUR>: Turkish <CZE>: Czech <IND>: Indonesian <VIE>: Vietnamese	
(3) NC serial No.	This displays the NC model name, serial No, system type, and unit type.		
	NC TYPE:	NC type	
	MODEL NAME:	System type name	
	SERIAL NO.:	Serial No.	
	UNIT NAME:	Unit type	

Display items	Details			Remarks
(4) Hardware list	The hardware names are displayed. See the next/previous page by using PAGE▲ / PAGE▼ keys, or ▲ / ▼ keys on the screen. The following display is an example.			▲ / ▼ keys appear only when the screen has items on the next/previous page.
	CNC	:WN125 :WN893 :WN094	Main card Back panel card 7SEG-RSW card Existing cards are displayed from the top in the above order.	A single unit of CNC unit is formed with a maximum of 4 PCBs. All of these cards are connected by bus.
	ATT CARD	:WN523 :	Front-side SD IF card	
	EXT	:WN561 :WN563 :WN564 :WN511	CC-Link expansion unit PROFIBUS-DP master unit CC-Link IE Field unit or EtherNet/IP unit (WN564 and WN511 are one set.) <Note> ♦For the expansion unit type, refer to the description of expansion unit. Existing expansion units are displayed from the top in the above order.	The extension unit is an optional specification of hardware. The back panel and maximum of 5 cards are displayed.
	DISPLAY	:DU193	Display unit	The unit type varies by model.
	KEYBOARD	:KB091	Keyboard unit	The unit type varies by the keyboard unit being connected.
	ECOMON1 to ECOMON16	:EMU4-BD1 to :EMU4-HD1	EcoMonitorLight unit	The unit type varies depending on the connected EcoMonitor Light unit. The status for up to 16 part systems is displayed.
	RIO1[n] RIO2[n] RIO3[n]	:WN301 :WN301 :WN301	Remote IO unit 1 (n = 1 to 64) Remote IO unit 2 (n = 1 to 64) Remote IO unit 3 (n = 1 to 64)	The unit type varies by the remote I/O unit being connected. Maximum of 3 channels are displayed.
(5) Project	The list of project Nos. is displayed. If the selected project is switched with [↑] or [↓] key, the list of ladder files that belong to the project is displayed.			Maximum of 10 single-byte alphanumeric characters for the label

Display items	Details		Remarks
(6) PLC ladder file list	This displays the file name, file label, execution type for each PLC ladder program. See the next/previous page by using PAGE▲/PAGE▼ keys, or ▲/▼ keys on the screen.		▲/▼ keys appear only when the screen has items on the next/previous page.
	(a) Registration No.	This displays the registration No. of each PLC program file.	The maximum number of projects to be registered in total is as follows: M800VW/M800VS: 120 M80VW/M80V: 60
	(b) File name	This displays the file name of PLC program file. (data of GX Developer/GX Works2)	Max. 8 characters
	(c) File label	This displays this file label of PLC program file. (data of GX Developer/GX Works2)	Max. 32 characters
	(d) Execution type	This displays the execution type of PLC program. HIGH: High-speed PLC program MIDDLE: Middle-speed PLC program INTIAL: Initial state of PLC program WAIT: Standby PLC program LOW: Low-speed PLC program (blank): Not the target of the execution.	

Menus

Menus	Details
S/W config	This displays the software list.
H/W config	This displays the hardware list.
Ladder config	This displays the PLC program list (file name, file label, and execution type).

11.2.2 Option Display Screen

Diagn – Option

Select the menu [Option] on the diagnosis (Diagn) screen to display the contents of the optional specifications registered in the NC.

The optional specification items are displayed by name. If all of the optional specifications cannot be seen in one screen, the rest of items can be displayed by pressing the page up/down keys.

Display items

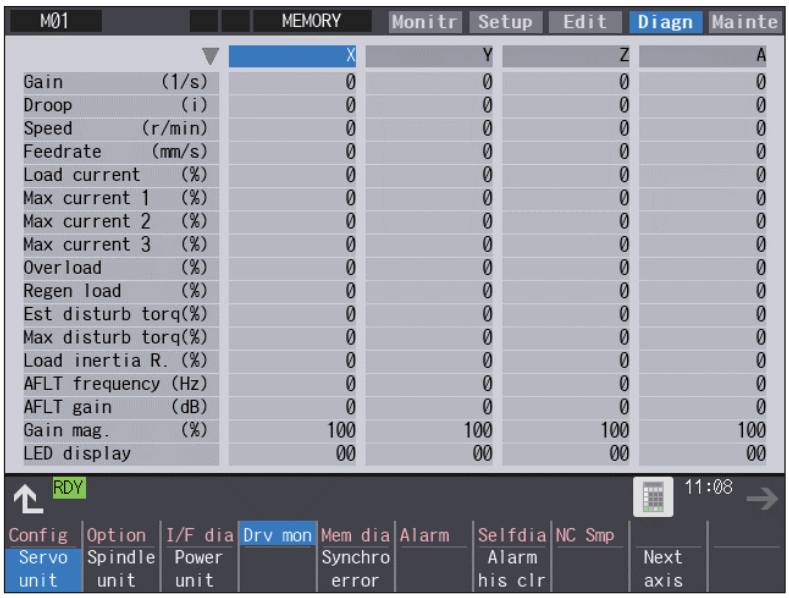
Display items	Details	Remarks
Option items	The list of currently usable optional specifications is displayed. As for the currently usable items, the background color is displayed in blue. The items set when the power supply was turned ON can be currently used. See the next/previous page by using PAGE▲/PAGE▼ keys, or ▲/▼ keys on the screen.	▲/▼ keys appear only when the screen has items on the next/previous page.

11.2.3 Drive Monitor Screen (Servo Unit)

Diagn – Drv mon

The various data related to the servo axis (NC axis, PLC axis) can be monitored by selecting the menus [Drv mon] -> [Servo unit] on the diagnosis (Diagn) screen.

Change the display item with PAGE / PAGE or ▲/▼ on the screen. ▲/▼ keys appear only when the screen has items on the next/previous page.



The axis name set in the base axis specification parameter "#1022 axname2" appears at the axis name.

Display items

Display items	Details
Gain (1/s)	This displays the position loop gain. The position loop gain is obtained by the following formula: $\frac{\text{Feedrate (mm/s)}}{\text{Tracking delay error (mm)}}$
Droop (i)	The error of the actual machine position to the command position is called droop. This error is proportional to command speed value. This follows the setting and display unit (#1003 iunit).
Speed (r/min)	This displays the actual rotation speed of motor. When a linear motor is used, the unit of feedrate (mm/s) is displayed.
Feedrate (mm/s)	This displays the feedrate on the machine side.
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.

Display items	Details
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 overload of motor and drive unit.
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 when the collision detection function is adjusted, in terms of a rate (%) to stall torque.
Max disturb torq (%)	This displays the maximum estimated disturbance torque (absolute value) when the collision detection function is adjusted, in terms of a rate (%) to stall torque. The 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. The ratio is displayed according to the parameter settings of "#1251 set23/bit0" and "#2235 V035/bitF".
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.
Gain mag. (%)	While Real-time tuning 1 is valid, the currently applied speed control gain is displayed with the magnification for the setting value of speed loop gain 1 (parameter "#2205 SP005 VGN1").
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 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 position	This displays the NC basic machine coordinate system position. (Command unit)
Motor end FB	This displays the feedback value of the motor side position detector. (Command unit)
Machine end FB	This displays the feedback position of the machine side position detector. (Command unit)
FB error (i)	This displays the error of the motor side FB and machine side FB. This follows the setting and display unit (#1003 iunit).
DFB compen amnt (i)	This displays the compensation pulse amount during dual feedback control. This follows the setting and display unit (#1003 iunit).
Remain command	The remaining movement distance of one block is displayed. (Command unit)
Current 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	The absolute position that does not include the machine error compensation amount is displayed. (Command unit)
Superimp syn er (mm)	This displays the current value of the synchronous error between the reference axis and the synchronized axis during control axis synchronization between part systems, control axis superimposition, or arbitrary axis superimposition.
Superimp err +P (mm)	This displays the maximum value of the distance in which the synchronized axis advances beyond the reference axis during control axis synchronization between part systems, control axis superimposition, or arbitrary axis superimposition.
Superimp err -P (mm)	This displays the maximum value of the distance in which the synchronized axis lags behind the reference axis during control axis synchronization between part systems, control axis superimposition, or arbitrary axis superimposition.

Display items	Details
Superimp errP-P (mm)	This displays the synchronous error width between the reference axis and the synchronized axis during control axis synchronization between part systems, control axis superimposition, or arbitrary axis superimposition.
Motor temp. (°C)	This displays the motor temperature of the servo motor. This item is not displayed when the parameter "#1760 cfgPR/bit4" is set to "1".
Power cycles (p)	This displays the power cycle counter. (Number of times)
Insul resist. (MOhm)	The insulation degradation status is displayed.
FAN1 rot. state (%)	The detected rotation speed of the unit cooling fan is displayed by converting it into percentage format (converted with the rated speed of the fan as 100%).
FAN2 rot. state (%)	The detected rotation speed of the unit cooling fan is displayed by converting it into percentage format (converted with the rated speed of the fan as 100%).
Battery voltage (V)	This displays the voltage of the connected battery.
AUX current sta No. (*1)	This displays the current station No.
AUX current posn (*1)	This displays the current coordinates position.
AUX inst station No. (*1)	This displays the command station No. of automatic operation or the nearest station No. to stop by manual operation etc.
AUX inst posn (*1)	This displays the coordinates position corresponding to target station No.
Mach err comp val	This displays the machine error compensation amount. (Command unit)
Rot/Spatial err comp	This displays the amount of rotation center error compensation and spatial error compensation.
Control input 1L	This indicates the control signal input from NC to servo. (bit0 to bit7) Bit0: READY ON command Bit1: Servo ON command Bit4: Position loop gain changeover command Bit6: Excessive error detection width changeover command Bit7: Alarm reset command
Control input 1H	This indicates the control signal input from NC to servo. (bit8 to bitF) Bit8: Current limit selection command
Control input 2L	This indicates the control signal input from NC to servo. (bit0 to bit7)
Control input 2H	This indicates the control signal input from NC to servo. (bit8 to bitF) Bit9: Speed monitor command valid BitA: In door closed (controller) BitB: In door closed (all drive units)
Control input 3L	This indicates the control signal input from NC to servo. (bit0 to bit7) Bit0: Control axis detachment command
Control input 3H	This indicates the control signal input from NC to servo. (bit8 to bitF)
Control input 4L	This indicates the control signal input from NC to servo. (bit0 to bit7)
Control input 4H	This indicates the control signal input from NC to servo. (bit8 to bitF)
Control input 5L	This indicates the control signal input from NC to servo. (bit0 to bit7)
Control input 5H	This indicates the control signal input from NC to servo. (bit8 to bitF)
Control input 6L	This indicates the control signal input from NC to servo. (bit0 to bit7) Bit0: OMR-FF control request
Control input 6H	This indicates the control signal input from NC to servo. (bit8 to bitF) Bit8: Drivers communication control request
Control output 1L	This indicates the control signal output from servo to NC. (bit0 to bit7) Bit0: In READY ON Bit1: In Servo ON Bit4: In position loop gain changeover Bit6: In excessive error detection width changeover Bit7: In alarm

Display items	Details
Control output 1H	This indicates the control signal output from servo to NC. (bit8 to bitF) Bit8: In current limit selection BitC: In-position BitD: In current limit BitE: In absolute position data loss BitF: In warning
Control output 2L	This indicates the control signal output from servo to NC. (bit0 to bit7) Bit0: Z-phase passed Bit3: In zero speed Bit7: In external emergency stop
Control output 2H	This indicates the control signal output from servo to NC. (bit8 to bitF) Bit9: In speed monitor BitA: In door closed (controller) BitB: In door closed (self drive unit)
Control output 3L	This indicates the control signal output from servo to NC. (bit0 to bit7) Bit0: In control axis detachment
Control output 3H	This indicates the control signal output from servo to NC. (bit8 to bitF)
Control output 4L	This indicates the control signal output from servo to NC. (bit0 to bit7)
Control output 4H	This indicates the control signal output from servo to NC. (bit8 to bitF)
Control output 5L	This indicates the control signal output from servo to NC. (bit0 to bit7)
Control output 5H	This indicates the control signal output from servo to NC. (bit8 to bitF)
Control output 6L	This indicates the control signal output from servo to NC. (bit0 to bit7) Bit0: In OMR-FF control
Control output 6H	This indicates the control signal output from servo to NC. (bit8 to bitF) Bit8: In drivers communication control
Detection system	This displays the detector type symbol of the absolute position detection system. ES: Semi-closed encoder EC: Ball screw side encoder LS: Linear scale MP: MP scale ESS: Semi-closed high-speed serial encoder ECS: Ball screw side 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.

Display items	Details
Control method	SEMI: Semi-closed loop CLOSED: Closed loop DUAL: Dual feedback
Motor end detector	This displays the motor side detector type.
Motor end detect No.	This displays the motor side detector serial No.
Machine end detector	This displays the machine side 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 side detector serial No.
Motor	This displays the motor type.
Work time	This displays the READY ON work time. (Units: 1 hr)
Alarm history 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 〜 Maint hist 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.

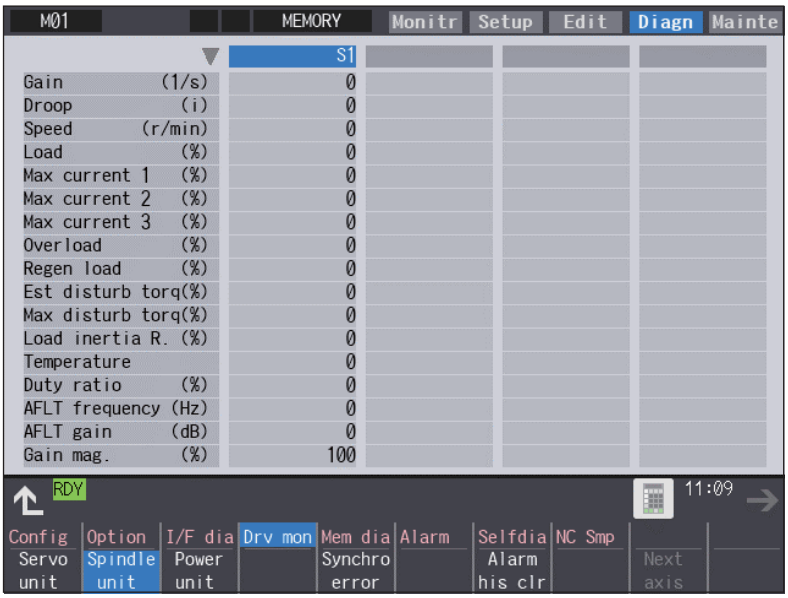
(*1) Data (station No./current position/command position) is displayed for axes of auxiliary axis state.
"-" is displayed for axes of non-auxiliary axis state (NC axis or during PLC axis control).

11.2.4 Drive Monitor Screen (Spindle Unit)

Diagn – Drv mon

The various data related to the spindle can be monitored by selecting the menus [Drv mon] -> [Spindle unit] on the diagnosis (Diagn) screen.

Change the display item with PAGE / PAGE or ▲/▼ on the screen. ▲/▼ keys appear only when the screen has items on the next/previous page.



Display items

Display items	Details
Gain (1/s)	This displays the position loop gain. The position loop gain is obtained by the following formula: $\frac{\text{Feedrate (mm/s)}}{\text{Tracking delay error (mm)}}$
Droop (i)	The error of the actual machine position to the command position is called droop. This error is proportional to command speed value. (*1)
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 overload of drive unit.
Regen load (%)	This is the data used to monitor the resistance overload state when the resistance regenerative power supply is connected.

Display items	Details
Est disturb torq (%)	This displays the estimated disturbance torque in terms of a rate (%) to rated torque when the spindle load monitoring function is enabled. In the coil changeover specification, it is displayed in terms of a rate (%) to rated torque when a high-speed coil is selected.
Max disturb torq (%)	This displays the maximum estimated disturbance torque (absolute value) in terms of a rate (%) to rated torque when the spindle load monitoring function is enabled. In the coil changeover specification, it is displayed in terms of a rate (%) to rated torque when a high-speed coil is selected. The maximum value remains displayed for 2 seconds.
Load inertia R. (%)	This displays the estimated load inertia ratio when the collision detection function is adjusted. The ratio is displayed according to the parameter setting of "#1251 set23/bit0".
Temperature (°C)	This displays the thermistor temperature. For the axis where the parameter "#43105 sp_temp_hide" (Hide Spindle temperature) is set to "0", the thermistor temperature is displayed according to the parameter "#1251 set23/bit1" and "#13225 SP225/bit2". For the axis where #43105 is set to "1", the thermistor temperature is hidden (blank).
Duty ratio (%)	This displays the equivalent load ratio of the spindle motor from cycle start of the program to the present.
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.
Gain mag. (%)	While Real-time tuning 1 is valid, the currently applied speed control gain is displayed with the magnification for the setting value of speed loop gain 1 (parameter "#13005 SP005 VGN1").
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 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) (*1)
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) (*1)
Machine position	This displays the NC basic machine coordinate system position. (Command unit) (*1)
Motor end FB	This displays the feedback value of the motor side position detector (Command unit) (*1)
FB error (i)	This displays the error of the motor side FB and machine side FB. (*1)
DFB compen amnt (i)	This displays the compensation pulse amount during dual feedback control. (*1)
Tap error (mm) (*2)	This displays the current value of the synchronous error between the tapping spindle and the drilling axis during synchronous tapping. (When the parameter "#1041 Initial inch" is set to "1", "Tap error (inch)" is displayed.) (*1)
Tap error +P (mm) (*2)	This displays the maximum value of the distance in which the drilling axis advances beyond the tapping spindle during synchronous tapping. (When the parameter "#1041 Initial inch" is set to "1", "Tap error +P (inch)" is displayed.) (*1)
Tap error -P (mm) (*2)	This displays the maximum value of the distance in which the drilling axis lags behind the tapping spindle during synchronous tapping. (When the parameter "#1041 Initial inch" is set to "1", "Tap error -P (inch)" is displayed.) (*1)
Tap error P-P (mm) (*2)	This displays the synchronous error width between the spindle and the drilling axis during the synchronous tapping. (When the parameter "#1041 Initial inch" is set to "1", "Tap error -P (inch)" is displayed.) (*1)
Tap error (deg) (*2)	This displays the current value of the synchronous error angle between the tapping spindle and the drilling axis during synchronous tapping. (±99999.999 deg) (*1)
Tap error +P (deg) (*2)	This displays the maximum value of the angle at which the tapping spindle advances beyond the drilling axis during synchronous tapping. (±99999.999 deg) (*1)

Display items	Details
Tap error -P (deg) (*2)	This displays the maximum value of the angle at which the tapping spindle lags behind the drilling axis during synchronous tapping. (±99999.999 deg) (*1)
Tap error P-P (deg) (*2)	This displays the synchronous error angle between the spindle and the drilling axis during the synchronous tapping. (deg) (*1)
SP sync error (deg)	This displays the current value of the synchronous error between the reference spindle and the synchronized spindle during spindle synchronization I, tool spindle synchronization IA/IB, or tool spindle synchronization II. (±99999.999 deg)
SP sync err +P (deg)	This displays the maximum value of the angle at which the synchronized spindle advances beyond the reference spindle during spindle synchronization I, tool spindle synchronization IA/IB, or tool spindle synchronization II. (±99999.999 deg)
SP sync err -P (deg)	This displays the maximum value of the angle at which the synchronized spindle lags behind the reference spindle during spindle synchronization I, tool spindle synchronization IA/IB, or tool spindle synchronization II. (±99999.999 deg)
SP syn err P-P (deg)	This displays the synchronous error width between the reference spindle and the synchronized spindle during spindle synchronization I, tool spindle synchronization IA/IB, or tool spindle synchronization II. (±99999.999 deg)
SP syn phas er (deg)	This displays the phase difference stored in the memory by calculating the phase shift value. (±99999.999 deg)
SP sync ph1 FB (deg)	This displays the current value of the feedback phase difference that includes the phase difference stored in the memory by calculating the phase shift value during spindle synchronization I, tool spindle synchronization IA/IB, or tool spindle synchronization II. (±99999.999 deg)
SP sync ph2 FB (deg)	This displays the current value of the feedback phase difference that does not include the phase difference stored in the memory by calculating the phase shift value during spindle synchronization I, tool spindle synchronization IA/IB, or tool spindle synchronization II. (±99999.999 deg)
SP syn er comp (deg)	This displays the error compensation amount during spindle synchronization I, tool spindle synchronization IA/IB, or tool spindle synchronization II. (±99999.999 deg)
GB sync error (deg)	This displays the current value of the synchronous error between the reference spindle and the guide bushing spindle during guide bushing spindle synchronization. (±99999.999 deg)
GB sync err +P (deg)	This displays the maximum value of the distance in which the guide bushing spindle advances beyond the reference spindle during guide bushing spindle synchronization. (±99999.999 deg)
GB sync err -P (deg)	This displays the maximum value of the distance in which the guide bushing spindle lags behind the reference spindle during guide bushing spindle synchronization. (±99999.999 deg)
GB sync err P-P (deg)	This displays the synchronous error width between the reference spindle and the guide bushing spindle during guide bushing spindle synchronization. (±99999.999 deg)
Motor temp. (°C)	This displays the motor temperature.
Power cycles (p)	This displays the power cycle counter. (Number of times)
Insul resist. (MOhm)	The insulation degradation status is displayed.
FAN1 rot. state (%)	The detected rotation speed of the unit cooling fan is displayed by converting it into percentage format (converted with the rated speed of the fan as 100%).
FAN2 rot. state (%)	The detected rotation speed of the unit cooling fan is displayed by converting it into percentage format (converted with the rated speed of the fan as 100%).
Battery voltage (V)	This displays the battery voltage.
Control input 1L	This indicates the control signal input from NC to spindle. (bit0 to bit7) Bit0: READY ON command Bit1: Servo ON command Bit7: Alarm reset command
Control input 1H	This indicates the control signal input from NC to spindle. (bit8 to bitF) Bit8 to BitA: Torque limit selection command
Control input 2L	This indicates the control signal input from NC to spindle. (bit0 to bit7)

Display items	Details
Control input 2H	This indicates the control signal input from NC to spindle. (bit8 to bitF) Bit9: Speed monitor command valid BitA: In door closed (controller) BitB: In door closed (all drive unit)
Control input 3L	This indicates the control signal input from NC to spindle. (bit0 to bit7)
Control input 3H	This indicates the control signal input from NC to spindle. (bit8 to bitF)
Control input 4L	This indicates the control signal input from NC to spindle. (bit0 to bit7) Bit0 to Bit2: Spindle control mode selection command Bit4: In gear changeover command Bit5 to Bit6: Gear selection command
Control input 4H	This indicates the control signal input from NC to spindle. (bit8 to bitF) BitC: M coil selection command BitD: L coil selection command BitE: Sub-motor selection command
Control input 5L	This indicates the control signal input from NC to spindle. (bit0 to bit7)
Control input 5H	This indicates the control signal input from NC to spindle. (bit8 to bitF) BitB: Minimum excitation 2 changeover request BitC: Speed gain set 2 changeover request BitD: Zero point re-detection request BitE: Increase holding power of spindle
Control input 6L	This indicates the control signal input from NC to spindle. (bit0 to bit7) Bit0: OMR-FF control request
Control input 6H	This indicates the control signal input from NC to spindle. (bit8 to bitF) Bit8: Drivers communication control request
Control output 1L	This indicates the control signal input from spindle to NC. (bit0 to bit7) Bit0: In READY ON Bit1: In Servo ON Bit7: In alarm
Control output 1H	This indicates the control signal input from spindle to NC. (bit8 to bitF) Bit8 to BitA: In torque limit selection BitC: In-position BitD: In torque limit BitF: In warning
Control output 2L	This indicates the control signal input from spindle to NC. (bit0 to bit7) Bit0: Z phase passed Bit3: In zero speed Bit7: In external emergency stop
Control output 2H	This indicates the control signal input from spindle to NC. (bit8 to bitF) Bit9: In speed monitor BitA: In door closed (controller) BitB: In door closed (self drive unit)
Control output 3L	This indicates the control signal input from spindle to NC. (bit0 to bit7)
Control output 3H	This indicates the control signal input from spindle to NC. (bit8 to bitF)
Control output 4L	This indicates the control signal input from spindle to NC. (bit0 to bit7) Bit0 to Bit2: In spindle control mode selection Bit4: In gear changeover command Bit5 to Bit6: In gear selection Bit7: Magnetic pole position not set
Control output 4H	This indicates the control signal input from spindle to NC. (bit8 to bitF) BitD: In L coil selection
Control output 5L	This indicates the control signal input from spindle to NC. (bit0 to bit7) Bit0: Current detection Bit1: Speed detection Bit6: In coil changeover

Display items	Details
Control output 5H	This indicates the control signal input from spindle to NC. (bit8 to bitF) Bit9: 2nd speed detection BitB: In minimum excitation rate 2 selection BitC: In speed gain set 2 selection BitD: Zero point re-detection complete BitF: In 2nd in-position
Control output 6L	This indicates the control signal input from spindle to NC. (bit0 to bit7) Bit0: In OMR-FF control
Control output 6H	This indicates the control signal input from spindle to NC. (bit8 to bitF) Bit8: In drivers communication control
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 side detector serial No.
Mach. end detect No	This displays the machine side detector serial No.
Motor	It displays the motor type which is set for the spindle specification parameter "#3138 motor_type". The maximum of 26 words are displayed with 2 lines and left aligned.
Work time	This displays the READY ON work time. (Units: 1 hr)
Alarm history 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 〜 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.

(*1) These follow the setting and display unit (#1003 iunit) of the 1st part system.

(*2) 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.

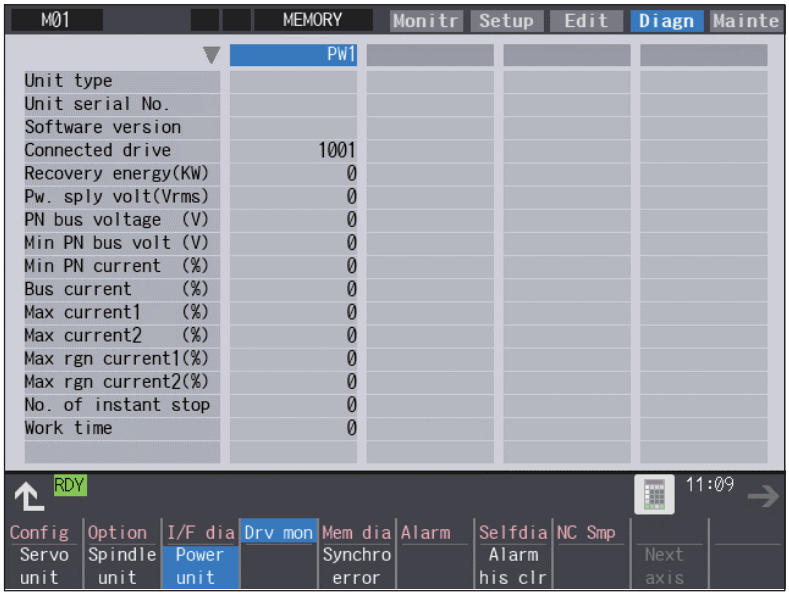
Data name	Details
Synchronous tapping error width (Max value)	♦This outputs the data of which absolute value is the largest among the synchronous tapping error width (-99999.999 to 99999.999 mm) during the synchronous tapping modal. ♦This data will be initialized to "0" when entering the synchronous tapping modal or restoring the power. Other than that, the data continues to display the maximum value.
Synchronous tapping error angle (Max value)	♦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.

11.2.5 Drive Monitor Screen (Power Supply Unit)

Diagn – Drv mon

The various data related to the power supply can be monitored by selecting the menus [Drv mon] -> [Power unit] on the diagnosis (Diagn) screen.

Change the display item with PAGE / PAGE or ▲/▼ on the screen. ▲/▼ keys appear only when the screen has items on the next/previous page.



Display items

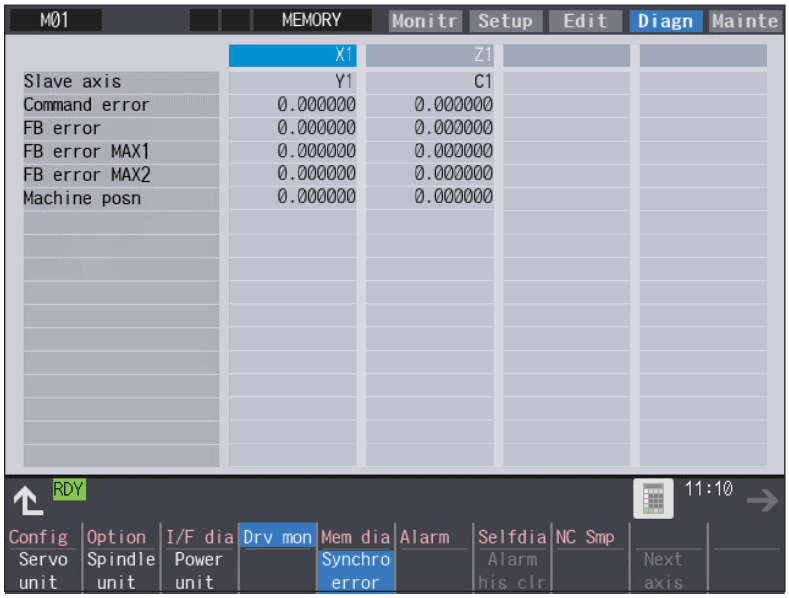
Display items	Details
Unit type	This displays the power supply unit type.
Unit serial No.	This displays the serial No. of the power supply unit.
Software version	This displays the software version.
Connected drive	This displays the I/F channel No. (mcp_no, smcp_no) of the drive unit connected to each power supply unit.
Recovery energy (KW)	This displays the regenerative power every two seconds. (0 to 999 kW)
Pw. sply volt (Vrms)	This displays the effective value of the power supply voltage. (0 to 999 Vrms)
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 power interruption exceeding 1 cycle of the power. (0 to 9999 times)
Work time	This displays the READY ON work time. (Units: 1 hr)
Power-run pwr (kW)	The power-run power is calculated and displayed. (0.01 kW)

Display items	Details
Regenerate pwr (kW)	The regenerated power is calculated and displayed. (0.01 kW)
Consumed power (kW)	The power consumption is calculated and its integrated value is displayed. (0.01 kWh)
Pwr. distortion (V)	The amount of power distortion (harmonic voltage value) is calculated and displayed. (V)
Supplied power (kW)	The supplied power is calculated and displayed. (kW)
Accum. power (kWh)	The accumulated power is calculated and displayed. (kWh)
Harmonic volt. (V)	The power line harmonic voltage is displayed. (V)
CV margin (%)	The CV margin is calculated and displayed. (%)
FAN1 rot. state (%)	The detected rotation speed of the unit cooling fan is displayed by converting it into percentage format (converted with the rated speed of the fan as 100%).
FAN2 rot. state (%)	The detected rotation speed of the unit cooling fan is displayed by converting it into percentage format (converted with the rated speed of the fan as 100%).
Alarm history 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 〜 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.

11.2.6 Drive Monitor Screen (Synchronous Error)

Diagn – Drv mon

The "Synchronous error" appears only when the synchronous control axis specification is valid.
The various data related to the synchronous error can be monitored by selecting the menus [Drv mon] -> [Synchro error] on the diagnosis (Diagn) screen.



Display items

Display items	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 position	This displays the commanded machine position for the master axis.

11.2.7 Clearing the Alarm History on Drive Monitor Screen

Diagn – 

Operation method

(1) Press the menu [Servo unit], [Spindle unit] or [Power 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.

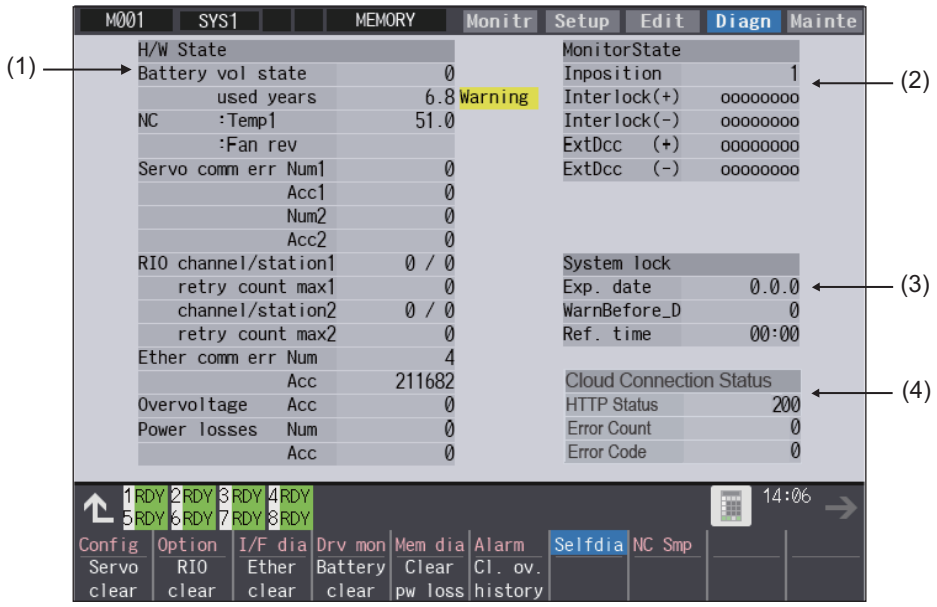
(4) Press the [Y] key. ➡ All alarm history data for the selected axis (device) are cleared to "0".

11.3 Diagnosis Screens

11.3.1 Self Diagnosis Screen

Diagn – Selfdia

The hardware state and NC operation state can be confirmed by selecting the menu [Selfdia] on the diagnosis (Diagn) screen.



Display items

	Display items	Details
(1)	H/W state (common for part systems)	This displays the hardware state of the NC unit and display unit. (*1)
(2)	Operation state (Depends on part system)	This displays the state when the operation seems to be stopped in spite that the alarm does not occur. (*2)
(3)	System lock	When the system lock specification is valid, the following items are displayed. Exp. date: The expiration date of the machine WarnBefore_D: The number of days to display the warning before the expiration date Ref. time: The time to display the warning or an alarm This will not displayed when the system lock specification is invalid. Once the expiry date is passed, cycle start is disabled. You can set a new expiration date or cancel the currently set expiration date by entering the decryption code issued by the MTB.
(4)	Cloud server connection status	Connection status of the cloud server is displayed when the remote service is enabled.

(*1) As for the NC unit, the contents are as follows depending on the condition:

NC

Item	Details	
Battery vol state	This displays the current state of the battery voltage as 0 or 1.	
	Condition	**Description**
	0 (normal state)	-
	1 (encoder error) (no battery)	Warning (yellow)

Item	Details	
Battery used years	This displays approximate time of the battery used from the last replacement.	
	Condition	**Description**
	Recommended battery use (5 years) ≤ Time for the battery used	Warning (yellow)
NC : temp. 1 (°C)	This displays the current temperature of the control unit.	
	Condition	**Description**
	M80V	
	82.0°C < Control unit temperature ≤ 84.5°C	Caution (gray)
	84.5°C < Control unit temperature	Warning (yellow)
	M800VS: Main card	
	82.0°C < Control unit temperature ≤ 84.5°C	Caution (gray)
	84.5°C < Control unit temperature	Warning (yellow)
	M800VW: Main card WN125A/WN115A	
	93.0°C < Control unit temperature ≤ 98.0°C	Caution (gray)
	98.0°C < Control unit temperature	Warning (yellow)
	M800VW: Main card WN125B or later/WN115B or later	
	79.5°C < Control unit temperature ≤ 87.0°C (*)	Caution (gray)
	87.0°C < Control unit temperature (*)	Warning (yellow)
	M80VW: Main card WN116A	
	97.0°C < Control unit temperature ≤ 102.0°C	Caution (gray)
	102.0°C < Control unit temperature	Warning (yellow)
	M80VW: Main card WN116B or later	
	79.5°C < Control unit temperature ≤ 87.0°C (*)	Caution (gray)
	87.0°C < Control unit temperature (*)	Warning (yellow)
Fan rev (r/min)	This displays the current fan rotation speed of the control unit.	
	Condition	**Description**
	Fan rot. speed ≤ 4000 r/min	Warning (yellow)

(*) The values listed in the table above are applicable for the software version A9 or later. If you are using the following version, the values differ as follows.

[M800VW: Main card WN125B or later/WN115B or later]

<Software version A7>

93.0°C < Control unit temperature ≤ 98.0°C Caution (gray)

98.0°C < Control unit temperature Warning (yellow)

<Software version A8>

83.0°C < Control unit temperature ≤ 85.0°C Caution (gray)

85.0°C < Control unit temperature Warning (yellow)

[M80VW: Main card WN116B or later]

<Software version A7>

97.0°C < Control unit temperature ≤ 102.0°C Caution (gray)

102.0°C < Control unit temperature Warning (yellow)

<Software version A8>

83.0°C < Control unit temperature ≤ 85.0°C Caution (gray)

85.0°C < Control unit temperature Warning (yellow)

Communication between NC unit and display unit

Item	Details
Servo comm err Num1	This displays the count of occurrence for "Y51 SV commu er: Recv frame No. xx04" after the power ON.
Servo comm err Acc1	This displays the cumulated count of occurrence for " Y51 SV commu er: Recv frame No. xx04". Press the menu [Servo clear] to clear the cumulated count to "0".
Servo comm err Num2	This displays the count of occurrence for "Y51 SV commu er: Data ID error xx03" after the power ON.
Servo comm err Acc2	This displays the cumulated count of occurrence for "Y51 SV commu er: Data ID error xx03". Press the menu [Servo clear] 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.
RIO retry count max 1	This displays the maximum value of the continuous error after the power ON.
RIO channel/station 2	This displays the Channel No./Station No. held even if the power OFF. Press the menu [RIO clear] to clear the Channel No./Station No. to "0/0".
RIO retry count max 2	This displays the count held even if the power OFF. Press the menu [RIO clear] to clear the count to "0".
Ether comm err Num	This displays the number of Ethernet communication error after PLC program is executed once.
Ether comm err Acc	This displays the cumulated count of occurrence for Ethernet communication error. Press the menu [Ether clear] to clear the cumulated count to "0".
Overvoltage Acc	This displays the cumulated count of detection frequency of overvoltage after the power ON. Press the menu [Cl. ov.history] to clear the cumulated count to "0".
Power losses Num	This displays the count of detection frequency of instantaneous power interruption after the power ON.
Power losses Acc	This displays the cumulated count of detection frequency of instantaneous power interruption after the power ON. Press the menu [Clear pw loss] to clear the cumulated count to "0".

(*2) The following state can be confirmed.

State	Details
In-position	This displays "1" (in-position state) when the following conditions are satisfied for even one axis. •There is a control axis whose acceleration/deceleration is not zero. •There is a control axis whose servo error exceeds the range designated with the parameter.
Interlock (+)	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) o o o o o o 1 o 8th axis … 1st axis 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.
Interlock (-)	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 (+)".

State	Details
ExtDcc (+)	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) o o o o o 1 o 1 8th axis … 1st axis 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.
ExtDcc (-)	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(+)".

Menus

Menus	Details
Servo clear	This clears the cumulated count of the servo communication error 1 and 2 to "0".
RIO clear	This clears the cumulated count of the RIO communication error to "0".
Ether clear	This clears the cumulated count of the Ether communication error to "0".
Battery clear	This clears the time the battery has been used to "0".
Clear pw loss	This clears the accumulated number of instantaneous power interruption detection times to "0".
Cl. ov. history	This clears the cumulated count of overvoltage to "0".

Clearing the cumulated counter to zero

(Example) Clearing the cumulated count of the servo communication error

(1) Press the menu [Servo clear]. ➡ An operation message appears.

(2) Press the [Y] or [INPUT] key. ➡ 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 menus [RIO clear], [Ether clear], [Battery clear], [Clear pw loss] and [Cl. ov.history].

Display of time limit warning and alarm

When designated date to display the expiration warning comes the warning message "Z64 Valid term soon to be expired" appears at the bottom right of the screen. At every cycle start after the NC power ON, this warning message appears until the designated warning-display time of the expiration date.

When the warning-display time has passed, the alarm message "Z65 Valid term has been expired" appears and cycle start cannot be performed. To cancel this alarm, enter the decryption code issued by MTB, and then turn OFF and ON the NC power.

Even when the expiration date has passed during the automatic operation, the NC continues to operate. However, after the NC once stops by cycle stop, single block stop or inputting a NC reset signal, cycle start cannot be performed.

For the display position of warning and alarm messages, refer to "2.1 Standard Screen".

While the warning is occurring, you can check the number of days remaining before the expiry date.
Open [Diagn] - [Alarm message] - [NC message] screen, then you can check the number of days. The remaining days will be "0" when the date reaches the expiration date.

If you open the [Alarm] screen while the alarm is occurring, "Valid term has been expired" appears. If you attempt a cycle start operation, the stop code "T01 Cycle st. prohibit (Term exp'd)" results, which disables cycle start operation.

Renewing time limit

For updating the time limit, users need to ask the MTB to issue a decryption code. The time limit is either updated or canceled when the decryption code is entered.
Decryption code is issued by MTB as decryption code file.

[How to enter decryption code]

(1) Save the decryption code file on an SD card (or USB memory).

(2) Insert the SD card into the control unit.

(3) Open the [Mainte] screen and select [Input/Output].

(4) In the [A:Dev] field, specify a memory card (or USB memory) in [Dev], a directory where the decryption code is stored in [Dir], and the created decryption code file name in [File].

(5) Select "Memory" in [B:Dev] and press [Dir]. Then the menu key [Decrypt code] appears on the screen. Then the menu key [Decrypt code] appears on the screen. When [Decrypt code] is pressed, "Decryption Code" automatically appears in [Dir] and "PASSCODE.DATA" in [File].
If you enter "/RLS" in [Dir], the screen will be displayed in the same way.

(6) Press [Trnsfr A-→B].

(7) When a confirmation message appears, press either [Y] or [INPUT].
If the decryption code has successfully been set, "Setting complete normally" appears.

- If the warning message is on the screen, this message will be removed.
- If the alarm message is on the screen, this message will be removed.
- If the decryption code is wrong, the message "Can't write file for dev B" will appear. Check the decryption code file and contact the MTB.
- The "Executing automatic operation" error occurs during program operation, and setting the code is disabled.

(8) Turn OFF and ON the NC power.

Precautions for expiry date

(1) If you turn OFF and ON the power after the time limit has expired, "EMG Emergency stop LINE" may appear at the same time.

(2) The warning message "Z64 Valid term soon to be expired" can be erased with NC reset. Contact the MTB for details.

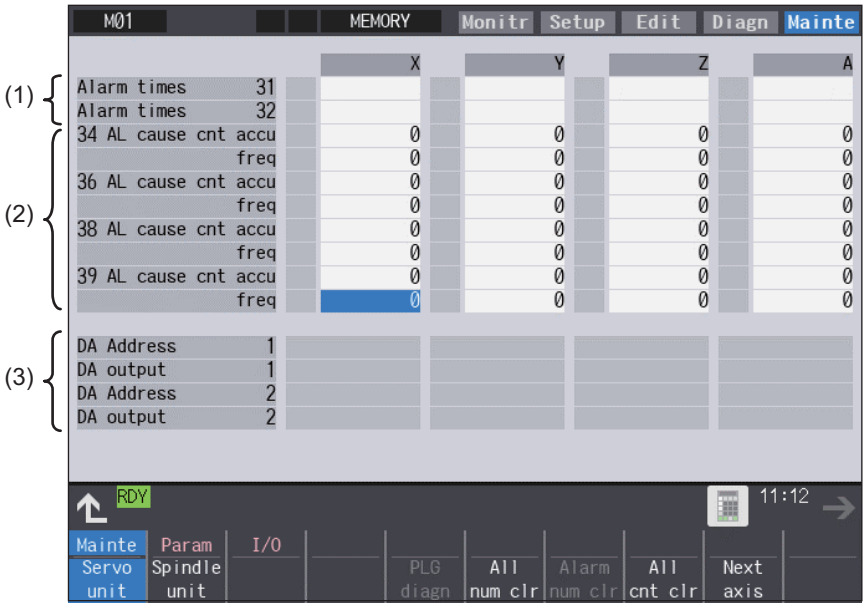
11.3.2 Servo Diagnosis Screen

Mainte – Mainte

You can confirm the following information on the servo diagnosis screen by pressing [Mainte] -> [Servo diagn] on the maintenance (Mainte) screen:
- The number of alarms of servo/spindle drive unit
- DA output information
- PLG diagnosis (spindle)
- The maintenance diagnosis information of servo/spindle unit such as alarm cause counter

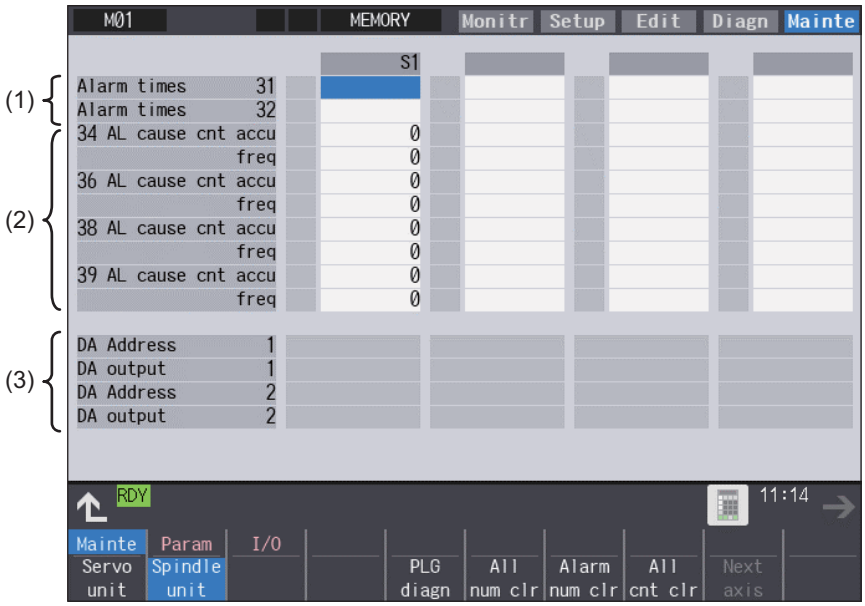
This operation is protected according to the operation level.

■Servo unit

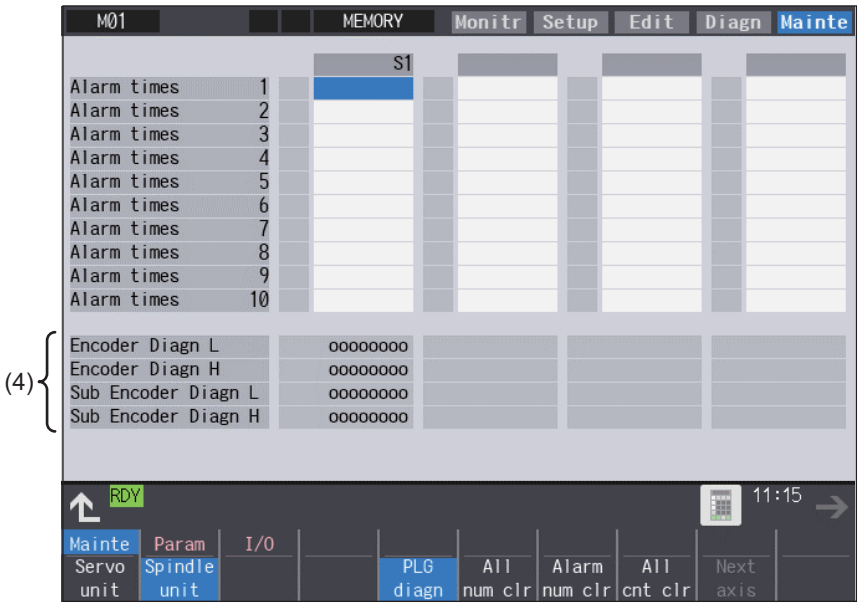


■Spindle unit

(1) Display of the DA output data



(2) Display of the spindle PLG diagnosis information



Display items

Display items	Details
(1) Alarm times	This displays the data for each axis or unit. There are Alarm times 1 to 32, and there are switched with the page up/down keys. The number of times can be cleared for each axis or each column. Alarm No: Servo alarm No. (unit: ASCII) No. of times: The number of times of meeting the servo alarm condition after the servo drive unit ON. (Units: Number of times)
(2) Alarm cause counter	An alarm occurs when the specified number of conditions are met. This counter displays the number of times in which the alarm occurrence condition is met since the drive unit power was turned ON. The number of times can be cleared for each axis with [All cnt clr]. accu: Displays the number of times in which the alarm occurrence condition was met since the drive unit power was turned ON. freq: Displays the peak hold value when the alarm occurrence condition is met per constant cycle.
(3) DA output	This displays the data for each axis or unit. DA Address 1: Servo parameter sv061 set value (address) (unit: ASCII) DA output 1: Servo parameter sv061 set address data (unit: pulse) DA Address 2: Servo parameter sv062 set value (address) (unit: ASCII) DA output 2: Servo parameter sv062 set address data (unit: pulse)
(4) Spindle PLG diagnosis information (Spindle drive unit only)	Displays the data for each axis Encoder Diagn L: Displays the output signal (Low) of the motor side PLG Encoder Diagn H: Displays the output signal (High) of the motor side PLG Sub Encoder Diagn L: Displays the output signal (Low) of the machine side PLG Sub Encoder Diagn H: Displays the output signal (High) of the machine side PLG

Note

(1) PLG (pulse generator) detects the pulse signal and it is used to detect the rotation state of the motor.

Menus

Menus	Details
Servo unit	Displays the servo unit diagnosis information in the data display area.
Spindle unit	Displays the spindle unit diagnosis information in the data display area. <Note> •When the parameter "#1039 spinno" is set to "0", this menu is not displayed.
PLG diagn	Switches the display between the DA output and the spindle PLG diagnosis information. Pressing this button again after this menu is highlighted, the display switches to the DA output. <Note> •When the parameter "#1039 spinno" is set to "0", this menu is not displayed. •Even when the parameter "#1039 spinno" is set to other than "0", this menu will be grayed out and non-selectable. while displaying the servo unit diagnosis information.
All num clr	This clears the number of alarms of the axis at the cursor position to "0".
Alarm num clr	This clears the number of alarms at the cursor position to "0". <Note> •When the cursor is on "AL cause cnt accu" or "freq", this menu is grayed out and non-selectable.
All cnt clr	Clears the all values of "AL cause cnt accu" and "freq" of the axis where the cursor is placed to 0.
Next axis	Displays the data of four axes from the next axis. This menu is displayed only when there are five or more servo axes or spindle.

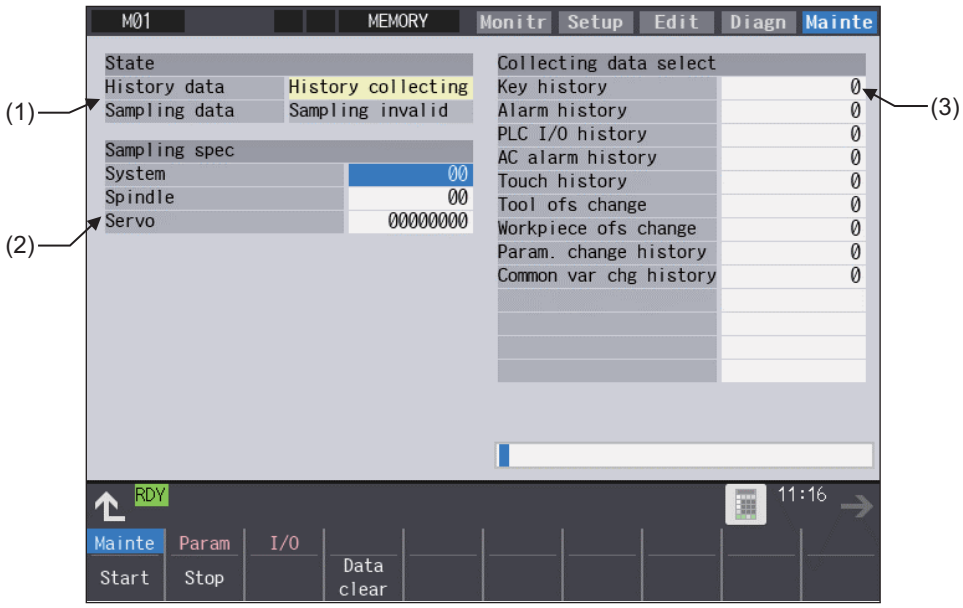
11.3.3 Collecting the Data on the Diagnosis Data Collection Setting Screen

Mainte – 

The NC automatically collects diagnosis data based on the "Sampling spec" and "Collecting data select" settings specified on the diagnosis data collection (Collect set) screen of the maintenance (Mainte) screen.
This operation is protected according to the operation level.

Note

(1) Make sure to use this function under our instructions. It may cause misoperation if you use the function based on your own judgments.



Display items

Display items		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.

Display items		Details
(2)	Sampling specifications	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. Specify the part system in HEX. 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. Specify the part system in HEX. 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. Specify the part system in HEX. 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.
(3)	Collecting data select	This selects the diagnosis data to be collected. 0: Collected. 1: Not 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 power ON/OFF history data is collected.
	Touch history	This selects whether or not touchscreen history data is collected.
	Tool ofs change	This selects whether or not tool offset change history data is collected.
	Workpiece ofs change	This selects whether or not workpiece offset change history data is collected.
	Param. change history	This selects whether or not parameter change history data is collected.
	Common var chg history	This selects whether or not common variable change history data is collected. Variable names are not included in the history data. <Note> ♦The change histories of extended common variables I/II (#10000 to #89999) and extended common variables III (#90000 to #97999) do not remain.

Menus

Menus	Details
Start	This begins collecting sampling data and history data.
Stop	This stops the sampling data and history data collection operation.
Data clear	This stops the sampling data and history data collection operation, and clears the collected data. This is not displayed when "#1263 bit1" is ON.

Note

(1) The alarm history data collection start/stop operation is switched using the menu [Start] or [Stop] of data collection regardless of whether "Alarm history" is set to "0" or "1".

Operation method (Start of data collection)

(1) Press the menu [Start].

(2) Press the [Y] or [INPUT] key.

Note

(1) Data collection starts even if "Alarm history" is set to "1" (not collected).

Operation method (Stop of data collection)

(1) Press the menu [Stop].

(2) Press the [Y] or [INPUT] key.

Note

(1) Data collection stops even if "Alarm history" is set to "1" (not collected).

Operation method (Clear of collected data)

(1) Press the menu [Data clear].

(2) Press the [Y] or [INPUT] key.

11.4 Advanced Diagnosis

Be sure to perform the operations in this section under the MTB instructions.

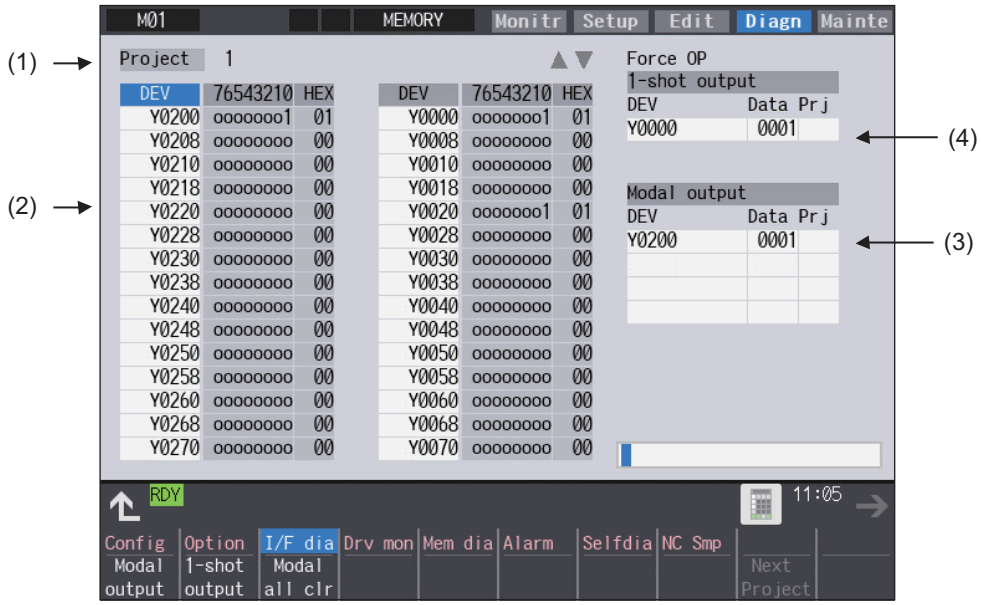
11.4.1 Displaying and Setting the PLC Device Data (I/F Diagnosis Screen)

Diagn – I/F dia

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

(1) Pay close attention to the sequence operation when using these functions during machine operation.



Display items

	Display items	Details
(1)	Project	This displays the currently displayed project.
(2)	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 display area with the [←] and [→] key when operations such as display changeover and data setting are carried out. Target device: X, Y, M, L, F, SB, B, SM, V, SW, SD, TI, TO, TS, TA, STI, STO, STS, STA, CI, CO, CS, CA, D, R, ZR, W The common devices for each project are X, Y, R, and ZR.
(3)	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. For details, refer to "Carrying Out Modal Output".

Display items	Details
(4) 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. For details, refer to "Carrying Out One-shot Output".

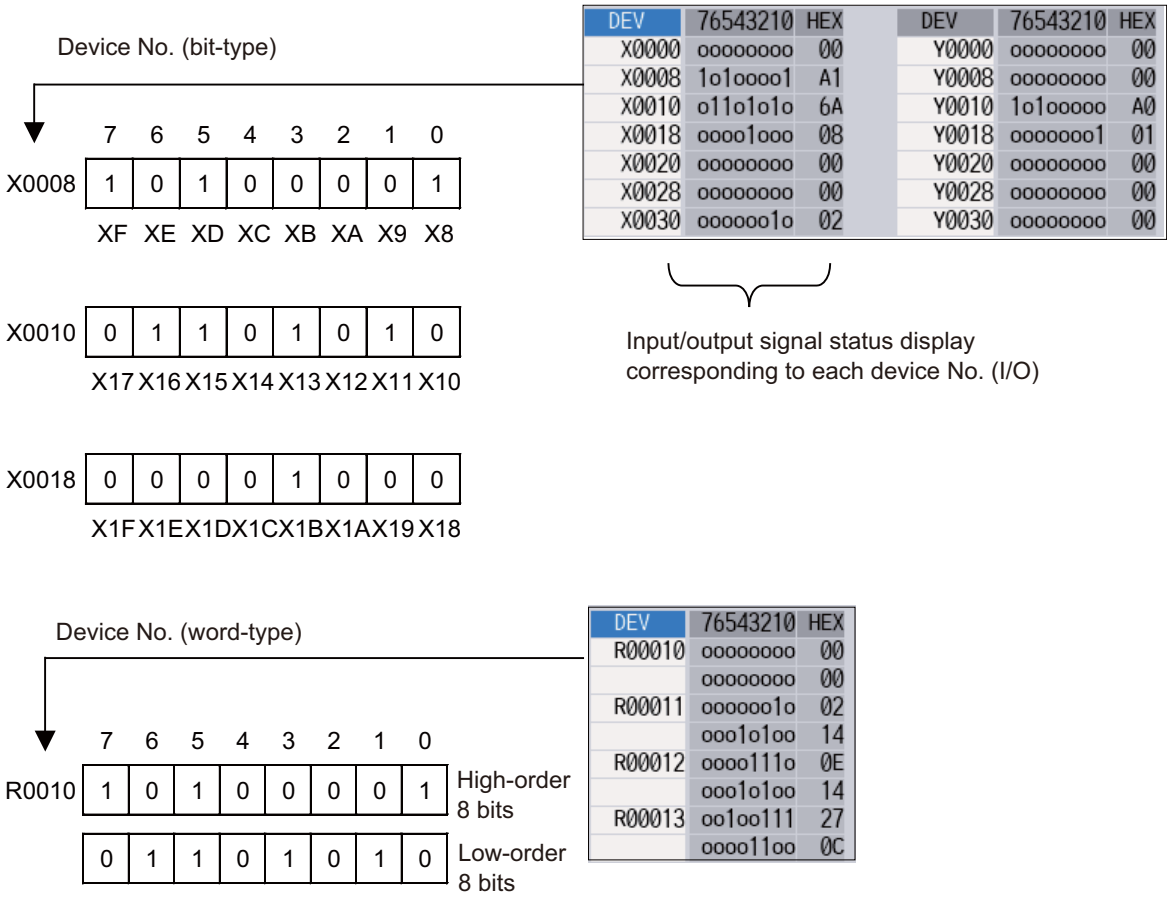
Menus

Menus	Details
Modal output	This changes the setting area to an input standby status. The signal is forcibly output (modal).
1-shot output	This changes the setting area to an input standby status. The signal is forcibly output (one-shot).
Modal all clr	A confirmation message appears, and the machine turns into standby status. The force OP (one-shot output) of all I/O signals are canceled. (*1)
Next Project	The device and I/O signal of the next project are displayed. If the last project is displayed, the top project is displayed.

(*1) These menus are grayed out ordinarily, and cannot be operated.

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.



11.4.1.1 Displaying the PLC Device Data

The various status signals and register data used in the PLC can be monitored and displayed.

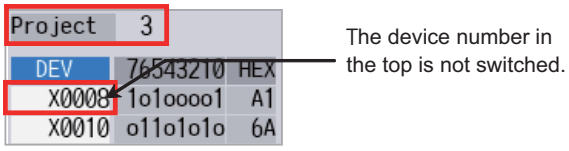
When this screen is first selected, the respective 16-byte amounts of input/output data are displayed from device "X0000" on the left display area, and from device "Y0000" on the right side.

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 [Next Project]. → The display is switched to the I/O signal of the next project. (Press the menu key to switch the project in order. Note that only projects are switched, but the device number currently displayed is not switched.)



(2) Press the menu tab key |←/→|, and select the area to display the data.

(3) Set the device No. (X0020), and press the [INPUT] key. → The device "X0020" appears at the head of the valid display area.

DEV	76543210	HEX	DEV	76543210	HEX
X0020	oooooooo	00	Y0000	oooooooo	00
X0028	oooooooo	00	Y0008	oooooooo	00
X0030	ooooo1o	02	Y0010	1o1ooooo	A0
X0038	oooooooo	00	Y0018	ooooooo1	01

Note

(1) When setting the device No., an error will occur if a number exceeding the specifications or an illegal address is set.
(2) The input/output signals of X, Y, R, and ZR are the same for all projects.

Changing the display with the page keys

The valid area device Nos. change in page units when PAGE/PAGE or ▲/▼ on the screen is pressed. Changing of the pages stops within the range of device numbers of which the device has. ▲/▼ keys appear only when the screen has items on the next/previous page.

(1) Press the PAGE key. → The data is displayed from the next number currently displayed.

11.4.1.2 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

Menus	Details
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.

Modally outputting data "1" to device "X0048"

(1) Press the menu [Modal output]. → The mode changes to the modal output mode, 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 [INPUT] key.
X0048/1 [INPUT] → 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 canceled by pressing the ◁ key or the ↵ key.

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 canceled or the power is turned OFF.
(2) When no data is set (Example: "X0048/","X0048"), the operation message "Setting Data not found" is displayed.
(3) The X, Y, R, and ZR devices are modally output for all projects.
(4) The devices other than X, Y, R, and ZR are modally output for the currently displayed project.

Releasing the modal output

(1) Press the menu [Modal output]. → The mode changes to the modal output mode, 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 [Modal clear]. → 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 canceled by pressing the ◁ key or 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.

11.4.1.3 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 [Next Project]. → The display is switched to the I/O signal of the next project.
(Press the menu key to switch the project in order.)

(2) Press the menu [1-shot output].

(3) Set the device and data, and press the [INPUT] key. → The input data is overwritten in the one-shot output area, and is one-shot output.
X0042/1 [INPUT]
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 canceled by pressing the key or 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.
(2) When no data is set (Example: "X0048/","X0048"), the operation message "Setting Data not found" is displayed.
(3) The X, Y, R, and ZR devices are one-shot output for all projects.
(4) The devices other than X, Y, R, and ZR are one-shot output for the currently displayed project.

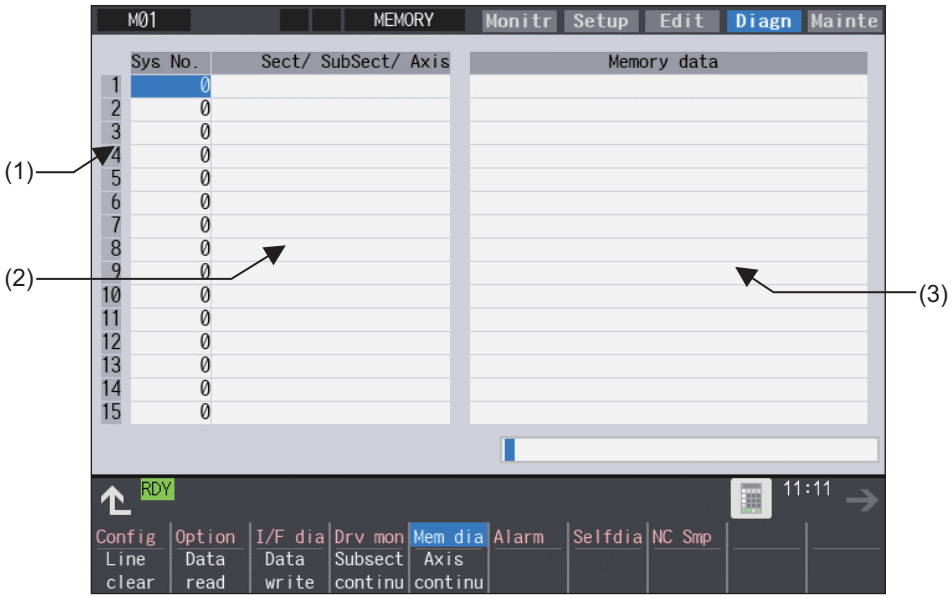
⚠ CAUTION

Pay close attention to the sequence operation when carrying out forced data setting (forced output) in the I/F diagnosis screen during machine operation.

11.4.2 Writing and Reading the Data Using the NC Data Designation (NC Memory Diagnosis Screen)

Diagn – Mem dia

The NC internal data can be displayed and rewritten on the NC memory diagnosis (Mem dia) screen of the diagnosis (Diagn) 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 items		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. (1: 1st part system). 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.

Menus

Menus	Details
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.
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 "Sys. No." of that line.
Data write	This writes the data in the setting area to the NC memory indicated by address data at the cursor position. (*1) The Data No. highlight (indicating data is being set) is released, and constant display is started. After writing, the cursor moves to "Sys. No." of the next line.
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 "Sys. No." of that line.
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 "Sys. No." of that line.

(*1) 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

11.4.3 Collecting the NC Data (Data Sampling Screen)

Diagn – 

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 on the maintenance (Mainte) screen.

Sampling specifications

Sampling cycle	1.776 ms x Setting value
Number of sampled axes	NC axis: Maximum number of usable axes Spindle: Maximum number of usable spindles
Number of sampled channels	1 to 16 points
Number of sampled data items	Max. 2,620,416 points (M800VW/M800VS Series) Max. 655,360 points (M80VW/M80V Series)

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.

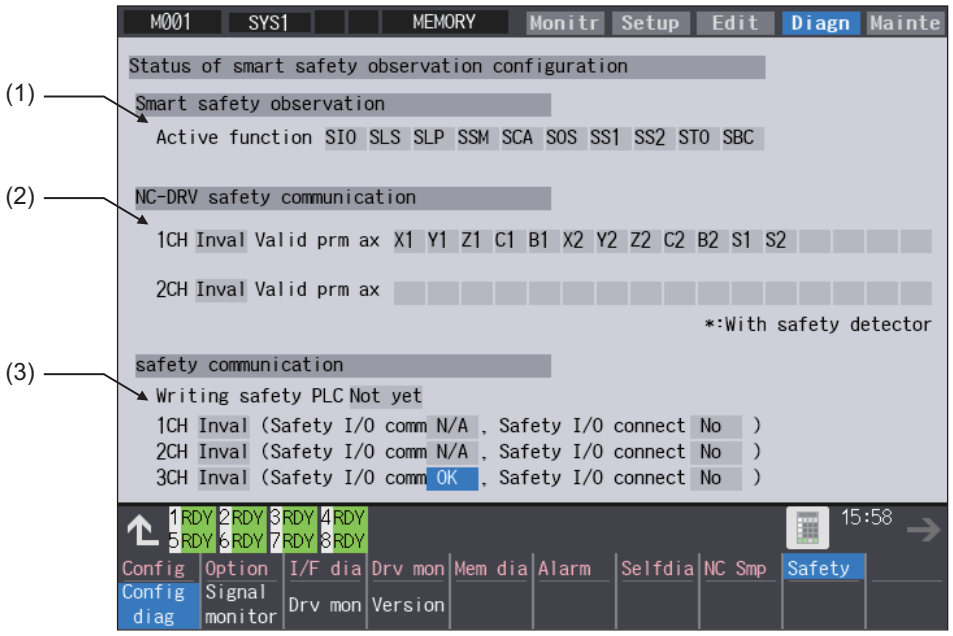
11.5 Safety Observation

Diagn – Safety

The configuration, signal status, drive status and version in use of the smart safety observation function are displayed on the safety observation screen.

11.5.1 Safety Observation Screen (Configuration Diagnosis)

The available/unavailable states of the smart safety observation function are displayed on the configuration diagnosis screen.



Display items

Display items		Details
(1)	Active function	The drive safety functions which are enabled with the parameters are highlighted. SIO: Safety-related I/O observation, emergency stop observation SLS: Safety-limited speed observation SLP: Safety-limited position observation SSM: Safe speed monitor SCA: Safe cam SOS: Safe operating stop observation SS1: Safe stop 1 SS2: Safe stop 2 STO: Safe torque OFF SBC: Safe brake control
(2)	NC-DRV safety communi-cation	This displays the safety communication status of the drive unit. To perform the safety drive communication, the smart safety observation target axis needs to be set by the parameter, and the connected channel with the drive unit needs to be optical communication and configured only with drive units MDS-E Series. Valid prm ax: Out of the axes connected to each channel, the axes that the parameter "#51101 SF_Disable" (Disable smart safety observation) is OFF are highlighted. For the axis with safety encoder, "*" appears at the top of the axis name.

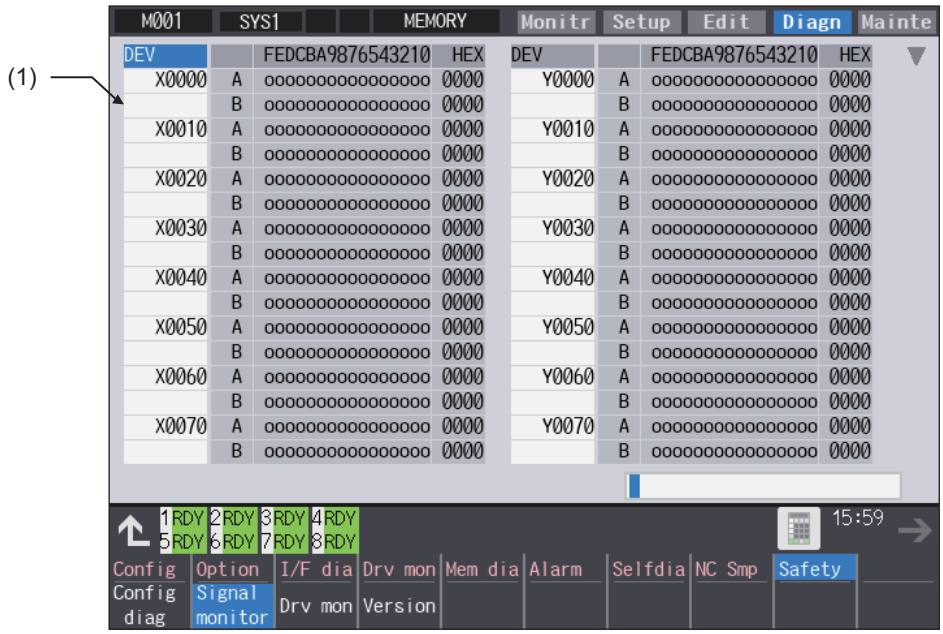
Display items	Details
(3) NC-I/O safety communication	This displays the communication status of the safety I/O communication. To perform the safety I/O communication, at least one of remote I/O channels needs to be configured with the safety I/O devices (safety remote I/O unit, operation panel equipped with safety I/Os) and RIO2.0 unit alone. Writing safety PLC: This displays the status (Done/Not yet) whether the PLC ladder for safety is written. Safety I/O comm: "OK" is displayed when the channel is configured only with RIO2.0 units. Safety I/O connect: "Yes" is displayed when connected to the safety I/O devices (safety RIO unit, operation panel equipped with safety I/O).

Menus

Menus	Details
Config diag	This switches the screen to the configuration diagnosis display of the smart safety observation.
Signal monitor	This switches the screen to the signal monitor display of the smart safety observation.
Drive monitor	This switches the screen to the drive monitor display of the smart safety observation.
Version	This switches the screen to the version display of the smart safety observation.
Servo unit	This switches the drive monitor diagnosis information to the servo unit information. Only when the drive monitor is displayed, the menu appears, and the operation can be performed.
Spindle unit	This switches the drive monitor diagnosis information to the spindle unit information. Only when the drive monitor is displayed, the menu appears, and the operation can be performed.
Next axis	This switches the axis to be displayed on the drive monitor. Only when the number of axes is five or more on the drive monitor screen, the menu appears, and the operation can be performed.

11.5.2 Safety Observation Screen (Signal Monitor)

The signal monitor screen displays the safety PLC signal status. The safety PLC signal displays both of the duplex PLC signals to identify the incongruous state. With the initial display, the input/output signal status from the device "X0000" is displayed on the left side and the input/output signals from the device "Y0000" is displayed on the right side.
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.



Display items

Display items		Details
(1)	Device No. and input/output signal value (binary/hexadecimal display)	This displays the duplex safety PLC signals (A, B) from the designated device No. in order. The input/output signal is displayed in binary and hexadecimal (bitwise). The device to be displayed can be switched individually for the left area and right area. The operation target (right or left) can be switched with the |← and →| keys. The head device and the operation target area are held until the power OFF. The displayable devices are X, Y, and ZR. If there is a comparison error with duplex safety PLC signals, the background color becomes yellow.

List of displayable PLC devices

Devices	Device No.	No. of points	Units	Details
X	X0 to X1FF	512	1-bit	Input signals to PLC, machine input, etc.
Y	Y0 to Y1FF	512	1-bit	Output signals from PLC, machine output, etc.
ZR	ZR0 to ZR3071	3072	16-bit	File register, CNC word I/F

Displaying the PLC device input/output signal

To refer the arbitrary input/output signal status, enter the target device No. to the input area, and the display will be switched.

(Example) For displaying the input/output signal of the arbitrary device No. "X0020"

(1) Press the tab key |← or →| to select the area where the data is displayed.

(2) Set the device No. (X0020), and then press the [INPUT] key.

Note

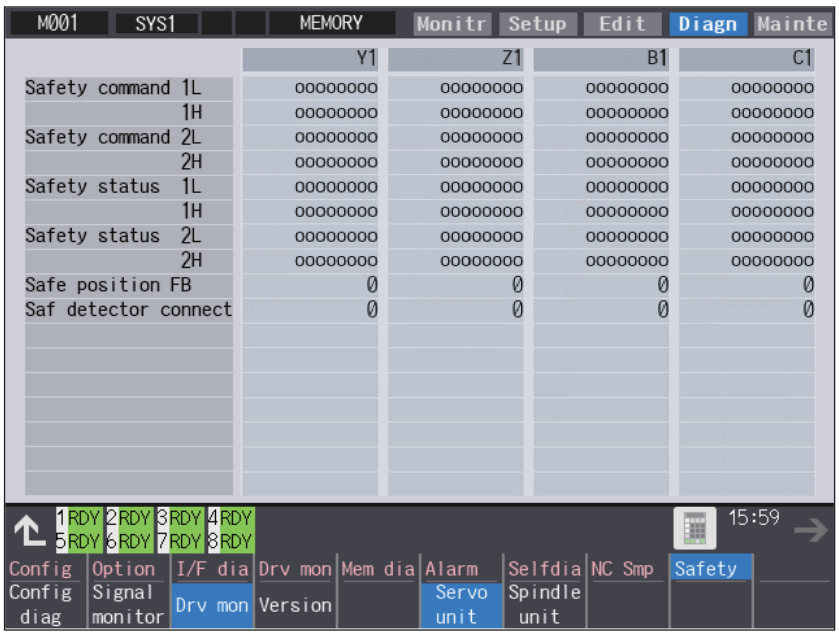
(1) Setting a device No. which exceeds the specification or setting an illegal address causes a setting error.

Changing the display with the page keys

The operation target device Nos. change in page unit when PAGE/PAGE or ▲/▼ on the screen is pressed. Changing of the pages stops within the range of device numbers of which the device has. ▲/▼ keys appear only when the screen has items on the next/previous page.

11.5.3 Safety Observation Screen (Drive Monitor)

The safety communication state with servo drive (NC axis, PLC axis) and spindle drive can be monitored. Only the safety drive communication parameter enabled axes are displayed on this screen.



Display items

Display items	Details
Safety command 1L Safety command 1H	This displays the data sent to the drive.

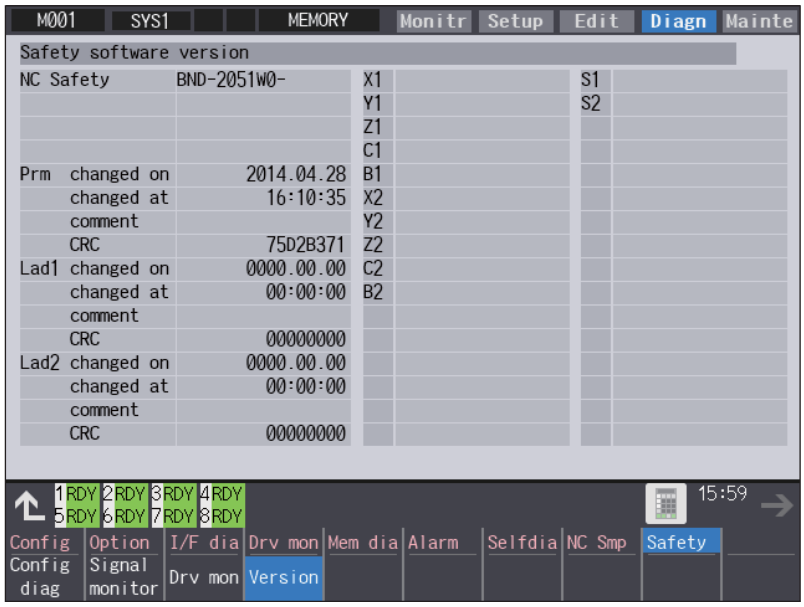
	Bit	Details
1L	0	STO command (0: No command, 1: With command)
	1	
	2	
	3	
	4	
	5	
	6	
	7	
1H	8	
	9	
	A	
	B	
	C	
	D	
	E	
	F	

<table>
<tr><th>Display items</th><th>Details</th></tr>
<tr><td>Safety command 2L</td><td rowspan="2">This displays the data sent to the drive.
<table>
<tr><th></th><th>Bit</th><th>Details</th></tr>
<tr><td rowspan="8">2L</td><td>0</td><td></td></tr>
<tr><td>1</td><td></td></tr>
<tr><td>2</td><td>SBC command (0: No command, 1: With command)</td></tr>
<tr><td>3</td><td>SBC command for SBT (1CH side) (0: No command, 1: With command)</td></tr>
<tr><td>4</td><td>SBC command for SBT (2CH side) (0: No command, 1: With command)</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></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>
</table></td></tr>
<tr><td>Safety command 2H</td></tr>
<tr><td>Safety status 1L</td><td rowspan="2">This displays the data received from the drive.
<table>
<tr><th></th><th>Bit</th><th>Details</th></tr>
<tr><td rowspan="8">1L</td><td>0</td><td>STO status (0: No operation, 1: In operation)</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">1H</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>
</table></td></tr>
<tr><td>Safety status 1H</td></tr>
</table>

<table>
<tr><th>Display items</th><th>Details</th></tr>
<tr><td>Safety status 2L</td><td rowspan="2">This displays the data received from the drive.
<table>
<tr><th></th><th>Bit</th><th>Details</th></tr>
<tr><td rowspan="8">2L</td><td>0</td><td></td></tr>
<tr><td>1</td><td></td></tr>
<tr><td>2</td><td>SBC status (0: No operation, 1: In operation)</td></tr>
<tr><td>3</td><td>SBC status for SBT (1CH side) (0: No operation, 1: In operation)</td></tr>
<tr><td>4</td><td>SBC status for SBT (2CH side) (0: No operation, 1: In operation)</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></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>
</table></td></tr>
<tr><td>Safety status 2H</td></tr>
<tr><td>Safe position FB</td><td>This displays the encoder position information from the drive to NC.</td></tr>
<tr><td>Saf detector connect</td><td>This displays the safety encoder connection status.
0: No connection
1: With connection</td></tr>
<tr><td>Safety status 2L</td><td>This displays the data received from the drive.</td></tr>
</table>

11.5.4 Safety Observation Screen (Version Display)

Software version about smart safety observation is displayed on the version display screen.



Display items

Display items	Details
NC Safety	This displays the smart safety observation software version of the NC system.
Parameters	This displays the safety parameter changed date and time, comment and CRC.
Lad 1, 2	This displays the safety PLC ladder 1, 2 changed date and time, comment and CRC.
Servo axis (up to 32 axes)	This displays the safety software version of servo axis. The version only for axis of the safety servo unit is displayed. Change the display item with PAGE / PAGE or ▲/▼ on the screen. ▲/▼ keys appear only when the screen has items on the next/previous page.
Spindle (up to 8 spindles)	This displays the safety software version of spindle. The version only for spindle of the safety spindle unit is displayed.

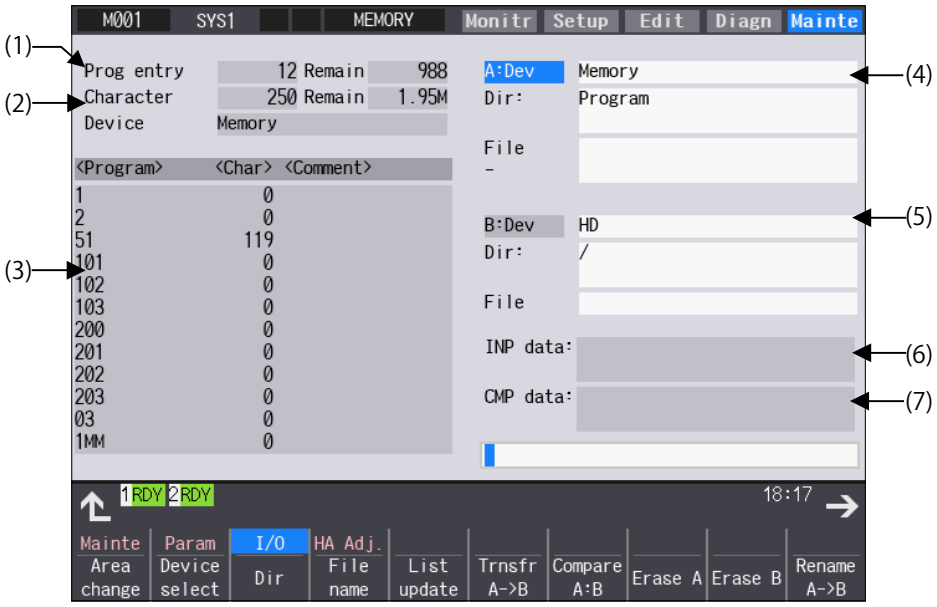
12

Performing a Backup Operation

Mainte – I/O

The Input/Output screen of the maintenance (Mainte) 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.

When the multi-part system program management is valid (#1285 ext21/bit0 =1) and the selected device is the NC memory, a program input and output will be executed across all part systems in batch if a parameter "#1285 ext21/bit2" (Multi-part system program management) is set to "0". When it is set to "1", the program input and output will be executed for the program of displayed part system.
However, when the machining program is transferred from NC memory or NC memory 2 to the other device, the transfer will be executed across all part systems in batch if "#1286 ext22/bit5" (Selection of multi-part system program input/output method) is set to "0". If it is set to "1", only displayed part system's program will be transferred.



Display items

Display items	Details
(1) Number of programs registered and remainder (*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. Remain: This displays the remaining number of programs that can be registered when NC memory or NC memory 2 is selected as a 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 (*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. Remain: This displays the remaining number of characters that can be registered. When NC memory or NC memory 2 is selected as a device, 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 (*2)	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. <Program> When NC memory or NC memory 2 is selected as a 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 NC memory or NC memory 2 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 the maximum (12 characters with the comment column, 29 characters without the comment column), the excess is indicated as "*". <Char> This displays the size of each file. When NC memory or NC memory 2 is selected as a device, this displays the number of characters of the machining program. When directory is selected, this displays "DIR". <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, memory card, DS, USB memory or Ethernet. For NC memory or NC memory 2, the display can be switched as follows depending on the parameter "#8981 NCmemory date disp" setting: (*3) When #8981 is set to "0", the machining program comment is displayed. When #8981 is set to "1", the file update date/time is displayed. When the comment of the NC memory or NC memory 2 exceeds the number of display characters (17 characters), the exceeded part is not displayed. The excess is not abbreviated to "*".
(4) File setting column A (5) File setting column B	This sets the device, directory, and file name for which the transmission, compare, and erase functions are used. 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. As for the device and directory, the setting values are retained even when the power is turned OFF. (*4) When a medium is changed on a device where a USB memory device, etc. can be removed or inserted, the directory before the change is displayed. In this case, press the menu [Dir], input "/", then press the menu [From list] to move to the target directory.
(6) Input data	This displays the data being transferred.
(7) Comparison data	This displays the data being compared. If an error occurs while comparing data, the block for which the error occurred is displayed.

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

○ : Displayed × : Not displayed

Display item/Device	Memory/ Memory2	HD	Serial	Memory card	DS/USB memory	Ethernet
Number of programs registered	○	○	×	○	○	○
Remain	○	×	×	×	×	×
Number of memory characters	○	○	×	○	○	○*
Remain	○	○	×	○	○	×
List	○	○	×	○	○	○

* When the parameter "#97*1 Host n no total siz" is set to "1", the number of host memory characters will not appear.

(*2) When the device is serial, an empty list is displayed.

(*3) For the program list view of the NC memory or NC memory 2, the program comment is displayed when the multi-part system program management is valid (#1285 ext21/bit0 = 1), regardless of the parameter setting of "8981 NCmemory date disp".

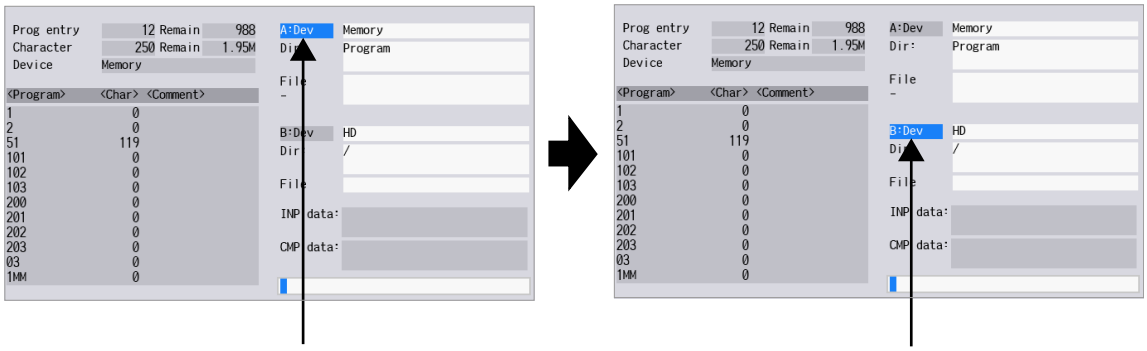
(*4) When the number of characters in the directory path (full path) from the root directory exceeds 100, the directory path is not retained, but a previously set directory path containing 100 characters or less retained. Maximum 37 characters can be entered as a file name.
When the device is NC memory or NC memory 2, the initial directory display is "Program" except when selecting directory from the menu.
When the device is Ethernet, the initial directory display is "/" (root).

Menus

Menus	Details
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.
Device select	This displays the sub-menu of the machining program storage area. When the sub-menu is selected, the device is confirmed, and if a directory exists it is set in the root. The memory is selected as the default.
Dir	This menu sets the directory that carries out input/ output operations, and is on standby for input. Note that when NC memory or NC memory 2 is selected as a device, the directory can be selected from the sub-menu.
File name	This menu sets the file name that carries out input/ output operations, and is on standby for input. When NC memory is selected as a device, setting is not necessary if the directory is not the program.
List update	This updates the list. The list of the directly selected in the currently valid file setting column (A/B) is updated.
Trnsfr 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. <Note> ・For Windows-based display unit of M800W/M80W, custom data cannot be transferred (input).
Compare A:B	This compares the files in file setting column A (transfer origin) and file setting column B (transfer destination). <Note> ・Custom data in NC memory cannot be compared.
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.
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.
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.
Comment nondisp	This changes whether to show or hide the comment field.
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, memory card or USB memory is selected for the device.
Merge A->B	This adds the contents of the file in file setting column B to that of in file setting column A. (The file in file setting column B is not changed.) <Note> ・The NC memory (excluding programs), serial and Ethernet (host file) cannot be merged.
MemCrd format	Format the memory card (front-side SD card).

Menus	Details
DS format	Format the memory card DS (built-in SD card).
USB Mem format	Format the USB memory.
Stop	This interrupts the process (transfer, compare, etc.) during its execution.

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.



File setting column A is valid.

File setting column B is valid.

Changing the valid file setting field

When file setting column A (top) is valid

(1) Press the menu [Area change]. → The file setting column B (bottom) is validated.

This can also be changed with the cursor key [↓].

12.1 Selecting a Device, Directory and File

Mainte – I/O

File selection sequence

Designate the device where the target file is located. → Select from the sub menu.

↓

Designate the directory with a full path. → Input the full path or select from the list.

↓

Designate the file name. → Input the file name or select from the list.

Menus

Sub-menus of [Device select]

Menus	Details
Memory	This selects the NC memory (program, parameter, PLC program, NC data).
Memory2	This selects NC memory 2.
HD	This selects the hard disk. This menu is not displayed on M800VS/M80V Series. (*1)
Serial	This selects the RS-232C device (PC, tape, etc.).
Memory card	This selects the front-side SD card.
DS	[M800VW/M80VW Series] This selects the SD card in the control unit. [M800VS/M80V Series] This selects the back-side SD card of the display unit.
Ethrnet	This selects the Ethernet-connected host computer.
USB Memory	This selects USB memory.

(*1) When the HD mode operation on an industrial computer (IPC) is enabled with M80V Series, the menu [HD] becomes selectable and you can select files in the hard disk on the industrial computer. The availability of this function depends on the MTB specifications (parameter "#1760 cfgPR10/bit2").
Set the IP address of the industrial computer to the parameter #11005 to use this function.

Sub-menus of [Dir] (when device is other than NC memory or NC memory 2) and [File name]

Menus	Details
From list	The cursor appears in the list display. The list contents can be selected with the [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] key is pressed again, it is fixed.
File name in	This copies the file name of the selected device to input area.
Clear filname	This deletes the file name of the selected device.
Top jump	[In normal mode] The contents of the first page in the list are displayed. [When importing data from the list] The contents of the first page in the list are displayed, and the cursor moves to the top row.
Bottom jump	[In normal mode] The contents of the last page in the list are displayed. [When importing data from the list] The contents of the last page in the list are displayed, and the cursor moves to the bottom row.

Sub-menus of [Dir] (when device is NC memory)

Menus	Details
Program	"Program" appears in the directory. The "File" column is blank.
Program all	"Program all" is displayed in the "Dir" column. The "File" column is blank.
Tool offset	"Tool offset" is displayed in the "Dir" column. "TOOL.OFS" is displayed in the "File" column.
Variabl	"Variable" is displayed in the Dir column. "COMMON.VAR" is displayed in the "File" column.
Coord offset	"Workpiece offset" is displayed in the "Dir" column. "WORK.OFS" is displayed in the "File" column.
All T data	"All tool data" is displayed in the "Dir" column. "TOOLALL.DAT" is displayed in the "File" column.
Param	"Parameter" is displayed in the "Dir" column. "ALL.PRM" is displayed in the "File" column.
System config	"System config" is displayed in the "Dir" column. "ASSEMBLY.INF" is displayed in the "File" column.
Safety param	"Safety parameter" is displayed in the "Dir" column. "SAFEPARA.BIN" is displayed in the "File" column.
Smp data	"Smp data" is displayed in the "Dir" column. "NCSAMP.CSV" is displayed in the "File" column.
Open SRAM	"Open SRAM" is displayed in the "Dir" column. "SRAMOPEN.DAT" is displayed in the "File" column.
Open device	"Open device" is displayed in the "Dir" column. "DEVOPEN.DAT" is displayed in the "File" column.

Menus	Details
WorkSet ErrComp	"WorkSetErrComp" is displayed in the "Dir" column. "WKPOS.OFS" is displayed in the "File" column.
Device Net	"DeviceNet" is displayed in the "Dir" column. "DEVICENT.PRM" is displayed in the "File" column.
Surface Data	"Surface Data" is displayed in the "Dir"column. "RNAVI.DAT" is displayed in the "File" column.
Interfr data	"Interference data" is displayed in the "Dir" column. "OBSTACL.DAT" is displayed in the "File" column.
System param	"System parameter" is displayed in the "Dir" column. "SYSTEM.PRM" is displayed in the "File" column.
Option	"Option" is displayed in the "Dir" column. "OPTION" is displayed in the "File" column.
Ladder	"Ladder" is displayed in the "Dir" column. "USERPLC.LAD" is displayed in the "File" column.
MCR	"MCR" is displayed in the "Dir" column. The "File" column is blank.
Email address	"Email transmission address" is displayed in the "Dir" column. "MAILADDR.DAT" is displayed in the "File" column.
Teach data	"Teach data" is displayed in the "Dir" column. "TEACHPOS.DAT" is displayed in the "File" column.
Lsr Cond	"Laser processing condition" is displayed in the "Dir"column. "LSRCOND.DAT" is displayed in the "File" column.
Extend com var	"Extended common variable" is displayed in the "Dir" column. "EXCOMMON.VAR" is displayed in the "File" column.
APLC data	"APLC data" is displayed in the "Dir" column. "APLC.BIN" is displayed in the "File" column.
Custom data	"Custom data" is displayed in the "Dir" column. The "File" column is blank.
HPU module	"HPU custom module" is displayed in the "Dir" column. The "File" column is blank.
HPU param	"HPU parameter" is displayed in the "Dir" column. "HPU_PARAM.BIN" is displayed in the "File" column.

Sub-menus of [Dir] (when device is NC memory 2)

Menus	Details
Program	"Program" appears in the directory. The "File" column is blank.
Program all	"Program all" is displayed in the "Dir" column. The "File" column is blank.

Selecting methods for device, directory, and file name

Device	Designation target file	Designation method		
		Device	Directory	File name
NC memory NC memory 2	♦Machining program ♦User macro ♦Fixed cycle	Select from the sub-menu	- (Default)	Key input in the input area, then press the [INPUT] key.
				Select from the list.
	Other than the machining program	Select from the sub-menu	Key input in the input area, then press the [INPUT] key.	- (Fixed)
			Select from the sub-menu	
Devices other than above	All	Select from the sub-menu	Key input in the input area, then press the [INPUT] key.	Key input in the input area, then press the [INPUT] key.
			Select from the list.	Select from the list.

Only machining programs can be stored on NC memory 2.

The device can be selected from the sub-menu. (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 or NC memory 2) and file name.

♦Designate the directory path (full path) or file name in the input area and press the [INPUT] key.

♦Press sub-menu [From list] of the menu [Dir] or [File name]. Move the cursor to the target directory or file name, and press the [INPUT] key.

A wildcard (*) can be used for the file name.

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 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.)

(3) 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 [INPUT] key again to fix the file name.

(4) If you press the [◁] key or the ← key when setting a file name, the file name in the input area is erased.

(5) When selecting a fixed cycle program, it is necessary to set the 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

Selecting a NC memory program

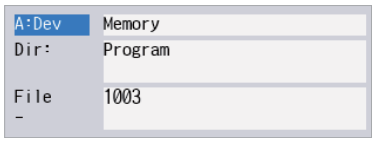
(1) Press the menu [Device select].

(2) Press the menu [Memory].

(3) Press the menu [File name].

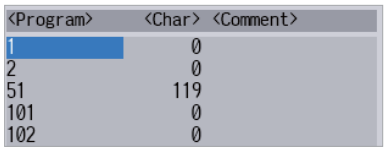
(4) Input the file name.
<When inputting the file name from the input area>
Input the file name.
10013 [INPUT]

→ The selected file name appears.



<When selecting the file name from the list>
(a) Press the menu [From list].

→ The cursor appears in the list.



(b) Move the cursor to the file name to be selected, and set.
[↑], [↓] [INPUT]

→ Selected file name is displayed in the input area.



(c) Press the [INPUT] key.

→ The selected file name appears.

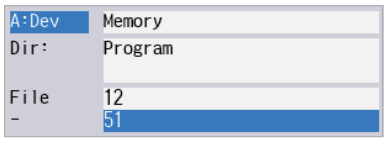
Designating multiple files

<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 name:
(Upper row) First file name
(Lower row) Last file name

→



<Using a wildcard>

A wildcard (*) can be used for the file name.

File: *
-

→ All files will be selected.

Note

(1) When serial or Anshin-net server is used, multiple files cannot be compared.

Selecting a file other than a program in NC memory

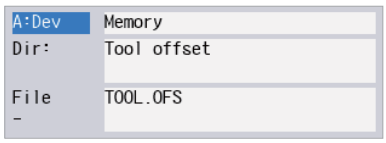
(1) Press the menu [Device select].

(2) Press the menu [Memory].

(3) Select the menu [Dir].

(4) Press the menu [Tool offset].

→ The directory and file name appear.



Note

(1) The file name for each directory is fixed. For the file name, refer to "12.8.2 List of File Names".

(2) When the menu [Program] or [Program all] is selected, the setting of "File name" column is cleared.

Selecting a device file other than the NC memory or NC memory 2 (When separately selecting the directory and file name.)

(1) Press the menu [Device select].

(2) Select the device.

(3) Select the menu [Dir].

(4) Select a file.

<When inputting the directory from the input area>

Input the directory path as a full path.

/PRG/PRECUT [INPUT]

<When selecting the directory from the list>

(a) Press the menu [From list].

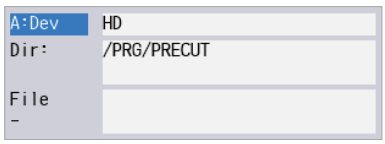
→ The cursor appears in the list.

(b) Move the cursor to the directory to be selected and set.

[↑], [↓] [INPUT]

Repeat this operation until the cursor arrives at the target directory.

→ The selected directory appears in the data setting column.
The contents of the selected directory appear in the list.



(c) When the target directory is reached, press the [◁] key or the ↵ key and quit the mode for inputting the directory.

Note

(1) Refer to next section "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 or NC memory 2 (When simultaneously selecting the directory and file name.)

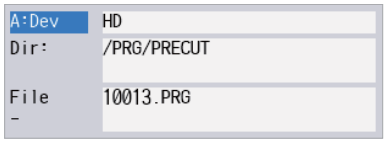
(1) Press the menu [Device select].

(2) Select the device.

(3) Press the menu [File name].

(4) Select a file.
<When inputting the file name from the input area>
Input the file name.
10013 [INPUT]

→ The selected file name appears.



<When selecting the file name from the list>
(a) Press the menu [From list].

→ The cursor appears in the list.

(b) Move the cursor to directory and file name to be selected, and set.
[↑], [↓] [INPUT]
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.

(c) Press the [INPUT] key.

→ The selected file name appears.

Canceling the input mode

(1) Press the [Device Select], [Dir] or [File name].

(2) Press the ◁ key or the ↵ key.

Changing or clearing the predefined file name

Using the menu [File name in] or [Clear filname] enables you to rename or clear the file name easily, which has already been set in "A:Dev." or "B:Dev".

■Changing a part of the predefined file name

(1) Select the device to rename the file, and press the menu [File name].

(2) Press the menu [File name in]. → The file name of the selected device is imported to the input area.

(3) Change the file name by using [→], [←], or [DELETE] key.

(4) Press the [INPUT] key.

■Clearing the predefined file name

(1) Select the device to clear the file name, and press the menu [File name].

(2) Press the menu [Clear filname]. → The file name is deleted.

12.2 Transferring a File

Mainte – 

Operation method

(1) Press the menu [Area change] to select file setting column A.

(2) Designate the transfer source 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 [Area change] to select file setting column B.

(4) Designate the transfer destination device, directory and file name.

(5) Press the menu [Trnsfr A->B].

(6) Press the [Y] or [INPUT] key.

⚠ CAUTION

⚠ " ; ", "EOB", "%", and "EOR" are symbols used for explanation. The actual codes for ISO are "CR, LF" ("LF") and "%". "%" and "EOR" are represented by "%" in ISO. The programs created on the edit screen are stored in the NC memory or NC memory 2 in a "CR, LF" format, however, the programs created with external devices may be stored in an "LF" format. The actual codes for EIA are "EOB (End of Block)" and "EOR (End of Record)".

To prevent influence from data omission and data transformation in the communication circuit, always verify the data after inputting and outputting machining programs.

Precautions

■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) When files are input to the NC memory or NC memory 2, or files in the said device are compared, if the file format size on the NC memory/NC momory 2 differs from the other side (i.e. the maximum number of registrations differs between the NC memory/NC memory 2 and the other side), processing is carried out matched to the smaller one.

(Example 1) If 200 files attempt to be input to an NC of 1000-file format, 200 files are registered.

(Example 2) If 1000 files attempt to be input to an NC of 200-file format, the files up to the 200th can be registered and an error message appears. (The remaining files are not registered.)

(4) Do not perform any file operation on the transfer source and destination files during the file transfer. The file may be damaged with the file operation.

(5) If the devices and the directories listed below are designated for the destination to transfer a file, a temporary file is created.
When the transfer is successfully completed, the name of the temporary file is changed to the designated one. (*1)
When the transfer is performed to overwrite an existing file, the temporary file is replaced with the existing one.

(1) [B:Dev]: Memory
Directory: Program, MCR (Motion control release), Field network configuration file

(2) [B:Dev]: DS, memory card (M800VS/M80V Series), USB memory (M800VS/M80V Series)

(*1) A temporary file remains in the following cases:

- When the transfer has terminated abnormally, for example, because the destination's capacity is insufficient.
- When the transfer has been interrupted.

The name of a temporary file starts with the symbol "~", and it consists of 11-digit numbers combined. The file can be deleted in the same manner as a normal file.

■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 execute 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 should be 250 characters or less.

(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

Up to 32 characters are allowed for the file name to create in or transfer to NC memory or NC memory 2.

(5) If the file whose name includes lower case alphabet is transferred to NC memory or NC memory 2, the lower case letter will be converted into the capital letter.

(6) When using tape, carry out parity V adjustment to improve the reliability of the tape format. Then use with the parameter "Parity V" validated.

(7) When the MTB 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

(8) Transferring or verifying the multiple files between the external device connected serially and that other than the serial connection.

(9) With machining program created before the MELDAS500 Series, "EOB" is registered as "LF". However, "EOB" will be converted to "CR LF", and the number of characters will increase in this Series. 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 this Series, the memory capacity may be exceeded.

(10) When the file to be transferred (input) is running or in "program restarting" mode, "program checking", the operation message "Executing automatic operation" or "Program restarting" or "Program checking" is displayed and file will not be transferred (input).

(11) If the file to be transferred (input) is being edited on the custom screen, the operation message "Can't write into being edited file" appears and may not be transferred (input).

(12) When the parameter "#8936 Delete leading 0" is set to "1", the file consisting only of numerical figures will be transferred with the "0" at the head deleted. For details, refer to "5.8.6 Leading Zero".

(13) When the multi-part system program management is valid (#1285 ext21/bit0 =1) and "#1285 ext21/bit2" (Multi-part system program generation and operation) is set to "0", the machining program's transfer to NC memory or NC memory 2 from the other device is executed across all part systems in batch. When the parameter is set to "1", only displayed part system's machining program will be transferred.

(14) When the multi-part system program management is valid (#1285 ext21/bit0 =1) and "#1286 ext22/bit5" (Selection of multi-part system program input/output method) is set to "0", the machining program's transfer from NC memory or NC memory 2 to the other device is executed across all part systems in batch. When the parameter is set to "1", only displayed part system's machining program will be transferred.

(15) The transfer range from the external device other than serial to NC memory or NC memory 2 will change depending on the value set in the parameter "#19006 EOR Disable".

<When the parameter "#19006 EOR disable" is set to "0">
The head line will not be transferred.
Up to "%" in the second line onward will be transferred, and data after "%" will not be transferred.
If there is no "%", all data will be transferred.

<When the parameter "#19006 EOR disable" is set to "1">
All file contents will be transferred.

(16) If the file to be transferred (input) is targeted for edit lock (edit lock B: 8000 to 9999 or edit lock C: 9000 to 9999), the message "Edit lock B" or "Edit lock C" is displayed, causing a transfer failure.
If the program name is input with only numbers when the parameter "#8936 Delete leading 0" is set to "1", the program is checked with the leading zero deleted. (Example: If the file name to be transferred (input) is designated by "008000" while edit lock B is enabled, the message "Edit lock B" is displayed, causing a transfer failure.)

(17) Multiple successive files or wildcard (*), if designated, is transferred, excluding the programs to be edit-locked, to suit the status of edit lock B or C.

(18) When the multi-part system program management is valid (#1285 ext21/bit0 =1) and a fixed cycle program is selected in NC memory, "Setting error" appears on inputting/outputting the program between NC memory and NC memory 2.

■Notes when transferring tool offset data files

(1) If an error occurs during offset data transfer, an error message appears on the screen, and the transfer operation is interrupted.

(2) Tool offset data file cannot be transferred from the serial to the memory.

■Notes when transferring all tool data files

(1) When tool life data is output from the NC memory, the tool information data, tool compensation data, and custom data allocation information are output per section.

Tool information data section: [TOOLINF_L1] (Tool life management I)
[TOOLINF_L2] (Tool life management II)
[TOOLINF_M] (Tool life management M system)

Tool compensation data section: [TOOLOFS]

Custom data allocation section: [CUSTOM]

(2) When all tool data is transferred to NC memory with either tool data or workpiece shift data protected, the message "Transfer skipping protected data?(Y/N)" appears. Pressing the [Y] key allows you to transfer the data with the protected data skipped. If both data are protected, the message "Data protect" appears and no data can be input.

(3) When all tool data is transferred to an external device from NC memory with either tool data or workpiece shift data protected, the message "Data protect" appears and the data cannot be output.

■Notes when transferring parameter files

(1) 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.

(2) 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 parameters again.

(3) When a protected parameter file (user parameter, machine parameter or pitch error compensation amount is protected) is transferred to NC memory, the message "Transfer skipping protected data?(Y/N)" appears. Pressing the [Y] key enables you to input the parameters except for protected data. If all parameters are protected, the message "Data protect" appears, no parameter can be input.

(4) When a protected parameter file (user parameter, machine parameter or pitch error compensation amount is protected) is transferred to an external device from NC memory, the message "Data protect" appears and the parameter cannot be output.

■**Notes when transferring common variable data files**

(1) 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.

(2) When the common variable protected from setting operation by the parameters #12111 to #12114, the message "Transfer skipping protected data?(Y/N)" appears. Pressing [Y] skips the protected variables, so they are not written to NC memory.

(3) When the common variable protected from setting operation by the parameters #12111 to #12114, the protected variables are not written to NC memory, even if the part system-based common variables included in the common variable data file are handled as the variables common to part systems by the NC.

■**Notes when transferring sampling data file**

(1) 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 1 m. When the output data length exceeds 1 m, lower 32 bits of the sampling data will be output.

(2) Sampling data file can be transferred from NC memory to an external device (output) but cannot be transferred from an external device to NC memory (input).

■**Notes when transferring workpiece offset data files**

(1) Tool offset data file cannot be transferred from the serial to the memory.

■**Notes when transferring machining surface data file**

(1) If you attempt to transfer (input) the data while a machining surface is selected in the machining surface selection screen, the error message "Can't write file for dev B" appears. In that case, cancel the selection of the machining surface.

■**Notes when transferring spatial error data file (POSTURE.DAT, POSTUREn.DAT (n: 2 to 4))**

(1) The spatial error data can be transferred from an external device to NC memory (input) but cannot be transferred from NC memory to an external device (output).

■**Notes when transferring email transmission log file**

(1) Log file can be transferred from NC memory to an external device (output), but it cannot be transferred from an external device to NC memory (input).

■**Notes when transferring teach data file**

(1) When the teach data file is protected, the message "Data protect" appears and the data cannot be input or output.

■**Notes when transferring motion control release file**

(1) The file can be transferred from an external device to NC memory (input), but it cannot be transferred from NC memory to an external device.

12.3 Comparing Files (Compare)

Mainte – 

Operation method

(1) Press the menu [Area change] to select file setting column A.

(2) Designate the device, directory and file name to be compared.

(3) Press the menu [Area change] to select file setting column B.

(4) Designate the other device, directory and file name to be compared.

(5) Press the menu [Compare A:B].

The file comparison starts.
The verifying data is displayed.
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.

Note

(1) Files that can be compared are text files only. Correct outcome will not be obtained through binary file comparison.

(2) Designating "*" for the compared file name causes "Setting error" when the comparison is executed with serial. Designate no file name or designate a target file name.

(3) "0" at the head of the file name will not be deleted when checking the individual file which consists of one program in one file even the parameter "#8936 Delete leading 0" is set to "1". Designating the file name is required. However, in the case of serial/batch file, comparison is carried out with a leading zero in the program name deleted.

(4) When multiple successive files or wildcard (*) are/is designated as the transfer source, the comparison is executed excluding the programs which are subject to the edit lock according to the status of Edit lock B and C.

(5) If you attempt the file comparison during the tool data sorting, the operation message "Can't open file for dev A (B)" appears, which disables the comparison.

12.4 Erasing a File

Mainte – I/O

Operation method (Erasing a file in file setting column A)

(1) Press the menu [Area change] to select file setting column A.

(2) Designate the device, directory and file name to be erased.

(3) Press the menu [Erase A].

(4) Press the [Y] or [INPUT] key.

➡ 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 menu [List update] to update the list.

Note

(1) 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.

(2) If the file to be erased is being edited on the custom screen, the operation message "Can't erase designated file" appears and may not be erased.

(3) When a wildcard "*" is designated for the file to be erased, the erasing is executed excluding the programs which are subject to the edit lock according to the status of Edit lock B and C.

(4) The image data associated with the file is also erased. However, both of the parameters "#1285 ext21/bit0" (Multi-part system program management) and "#1285 ext21/bit2" (Multi-part system program generation and operation) are set to "1", and when a file of the same name exists in the part system other than selected one, the image data associated with the file will not be erased.

Operation method (Erasing a file in file setting column 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].

12.5 Changing a File Name (Rename)

Mainte – I/O

Operation method

(1) Press the menu [Area change] to select file setting column A.

(2) Designate the original device, directory and file name.

(3) Press the menu [Area change] to select file setting column B.

(4) Designate the new device, directory and file name.

(5) Press the menu [Rename A->B].

(6) Press the [Y] or [INPUT] key.

Note

(1) Make sure the original and new devices are the same.

(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.

(3) If the file to be renamed is being edited on the custom screen, the operation message "Can't rename designated file" appears and may not be changed.

(4) 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.

(5) The directory where the file in operation search/check search is stored cannot be renamed. The operation message "Can't rename designated file" appears.

(6) The image data associated with the file is also renamed. However, both of the parameters "#1285 ext21/bit0" (Multi-part system program management) and "#1285 ext21/bit2" (Multi-part system program generation and operation) are set to "1", and when a file of the same name exists in the part system other than selected one, the image data associated with the file will not be renamed.

12.6 Creating a Directory

Mainte – 

Operation method

(1) Press the menu [Area change] to select the file setting column A.

(2) Designate the device.

→ The designated details appear.

The devices that can create a directory are HD, memory card, DS, and USB memory.

(3) Designate the directory where the new directory is to be created.

(4) Set the new directory in the file setting column A.

(5) Press the menu [Dir create].

The directory can also be created in file setting column B.

12.7 Restrictions

(1) For Ethernet input-output, you can insert only one LAN card into the PC side. If multiple LAN cards are inserted simultaneously, no operations are assured.

(2) When multiple files are transferred to NC memory or NC memory 2, files in edit lock mode are not transferred. All other files are transferred.

Device definition

(1) Removable disks in drive E and after are used as memory card devices. (Drive name advanced by the value of "#1390 BackUSBUseNum" from drive E. (Example): When #1390: 2, drive G is set as the drive name.) (*1)

(2) Removable disks in drive F and after are used as USB memory devices. (Drive name advanced by the value of "#1390 BackUSBUseNum" from drive F. (Example): When #1390: 2, drive H is set as the drive name.) (*1)

(3) If a USB memory device formatted over FAT32 is used, it may cause a file reading and writing failure.
Use a USB memory device that is formatted with FAT32.

(4) If a USB memory device is removed or inserted while data is being transferred to it, data integrity cannot be guaranteed. While message "Now, transferring data" or "Now, formatting data" is being displayed, do not remove or insert the USB memory device.
After message "Transfer completed" or "Formatting completed" has been displayed, remove or insert the USB memory device.

(*1) Only applies for the M800VW/M80VW Series.

List display

(1) If the number of full path characters exceeds 128 when the directory is being moved, it results in a movement failure.

(2) In the NC memory or NC memory 2, the cursor movement key and [INPUT] key cannot be used to navigate directories.

(3) The file name and directory name must be designated using less than 64 characters. If the file name or directory name is designated using 64 or more characters, it may not be recognized as the file name or directory name.

(4) If a directory that contains a double-byte code file name such as a Japanese file name is displayed in list view, such a double-byte code file name is not displayed correctly. In addition, the number of file characters, comments, and number of stored characters is not displayed correctly.

(5) In program list view on NC memory or NC memory 2, program comments are displayed when the multi-part system program management is enabled (#1285 ext21/bit0 = 1), regardless of the parameter setting of "#8981 NCmemory date disp".

(6) In program list view for Ethernet, if the host computer is unreachable, such as due to an incorrectly configured Ethernet parameter, the operation message "List updating" will be displayed until the connection times out, and no on-screen operations are possible.

(7) When NC Memory 2 is selected as a device, the only available directory settings are [Program] and [Program all]. If a directory is input, "Setting error" appears.

Program name

The following restrictions apply to file names that can be created and transferred in NC memory and NC memory 2.

(1) The file name must be designated using up to 32 characters, including an extension.

(2) Single-byte digits, single-byte uppercase alphabet characters, and Windows-recognizable single-byte symbols are available for the file name or directory name. Therefore, a double-byte code file name such as a Japanese file name cannot be used.

(3) Edit lock B, edit lock C, and program display lock are available only for files with names containing only single-byte digits in the NC memory or NC memory 2.

Example: When edit lock B (8000 to 9999) is enabled:

File name	Feature	Change
8000	Single-byte digits only	Not available
8000.PRG	With extension	Available
08000	Single-byte digits only, beginning with "0"	Available
8000A	With characters other than digits	Available

(4) The following names cannot be used as file names.

♦File name with extension "$$$", "$$0", "$$1", "$$2", "$$3", "$$4", "$$5", "$$6", "$$7", "$$8", or "$$9"

♦"0" (File named "single-byte zero")

(5) If NC memory is selected as a device, the number of characters allowed in the file name will depend on the setting for parameter " #1166 fixpro ".

Type	Setting value of #1166	Character limits, etc.
General program	0	32 characters
Fixed cycle program	1	9 digits, numbers only
MTB macro	MTB macro password	32 characters

If NC memory 2 is selected as a device, file names can be up to 32 characters, regardless of the setting for parameter "#1166 fixpro".

ALL.PRM input

(1) ALL.PRM data is assumed as a comment by adding semicolon (;) at the head of the data, and it can be excluded from being input. However, the message "Compare error" appears when it is compared.

(2) When ALL.PRM data which has header information (output date, CNC name, unit name, serial number and NC system information) is input to the older version device, the operation message "Can't write file for dev B" appears.

(3) Output format can be changed with the parameter "1218 aux02/bit3" (Parameter input/output format), but the format of header information is unchanged.

(4) When the parameter files (ALL.PRM) output from NC memory to the device (HD, DS or USB memory etc.) are compared, the message "Compare error" appears because of the discrepancy of the header information.

12.8 Other Functions

12.8.1 Merging a File

Mainte – I/O

Operation method

(1) Press the menu [Area change] to select file setting column A.

(2) Designate the merge destination device, directory, and file name.

(3) Press the menu [Area change] to select file setting column B.

(4) Designate the merge source device, directory, and file name.

(5) Press the menu [Merg B->A].

(6) Press the [Y] or [INPUT] key.

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.;
(FILE B)
G91 G28 XYZ
F1000;
G01 X200. Y200.;
M02;
%
```

Details of file in file setting column A before merge.

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 or in "program restarting" mode, the operation message "Executing automatic operation" or "Program restarting" is displayed, and the files will not be merged.

(2) If the merge destination file (file in file setting column A) is being edited on the custom screen, the operation message "Can't write into being edited file" appears and may not be merged.

(3) When the parameter "#19006 EOR Disable" is set to "1", a file is merged as it is. But the terminal character of the merge destination file (file in file setting column A) is "%", a line feed code is added and then merged.

(4) The devices that can be merged are NC memory, NC memory 2, HD (*1), memory card, DS and USB memory.

(5) Do not perform the file operation on the merge source and destination files during the merge. The file may be damaged with the file operation.

(6) When the multi-part system program management is valid (#1285 ext21/bit0 =1), merging files between NC memory and NC memory 2 is not available and "Setting error" appears.

(*1) Hard disk which is displayed by connecting with an industrial computer is excluded.

12.8.2 List of File Names

There is a directory for each type of data in NC memory and NC memory 2.

Each directory and file name (fixed) in the device is shown below.

NC memory 2 has the only directory for machining programs.
Do not change the extensions (.XXX) when storing in a device other than NC memory or NC memory 2.

Data type	NC memory directory path	Fixed file name
Machining program	/PRG/USER /PRG2/USER	(Program No.)
Fixed cycle program	/PRG/FIX	(Program No.)
MTB macro	/PRG/MMACRO	(Program No.)
User macro	/PRG/UMACRO	(Program No.)
Parameters	/PRM	
Parameters [User, machine] (Text format)		ALL.PRM
DeviceNet		DEVICENT.PRM
Safety parameter file		SAFEPARA.BIN
PLC program	/LAD	
PLC program file for each project		PROJECTn.LAD (n: 01 to Usable project No.)
Own station safety PLC program file		SAFEPLC1.LAD
Other station safety PLC program file		SAFEPLC2.LAD
NC data	/DAT	
Tool compensation amount data		TOOL.OFS
All tool data		TOOLALL.DAT
Common variable data		COMMON.VAR
SRAM data		SRAM.BIN
Workpiece offset data		WORK.OFS
Option file		OPTION-ADD.OPT
Workpiece installation error compensation data		WKPOS.OFS
SRAM open data		SRAMOPEN.DAT
Device open data		DEVOPEN.DAT
Machining surface data		RNAVI.DAT
Spatial error data file (text format)		POSTURE.DAT POSTUREn.DAT (n: 2 to 4)
Spatial error data file (binary format)		POSTURE.BIN POSTUREn.BIN (n: 1 to 4)
Interference data		OBSTACL.DAT
Email transmission setting file		MAILSET.DAT
Email transmission address file		MAILADDR.DAT
Teach data		TEACHPOS.DAT
Laser processing condition file		LSRCOND.DAT
Extended common variable data (Extended common variable I/II)		EXCOMMON.VAR
System configuration data	/DGN	ASSEMBLY.INF
Decryption code	/RLS	PASSCODE.DAT

Data type	NC memory directory path	Fixed file name
Sampling data (binary)	/LOG	NCSAMP.CSV
Email transmission log file		MAILSENDn.LOGN (n: 1 to 10)
History data		NCSAMP.BIN
All history		ALLLOG.LOG
Key history		KEYLOG.LOG
Alarm history		ALMLOG.LOG
Touch history		TOUCHLOG.LOG
Detailed alarm information history	/LOGEX	ALM-SVn.LOG
		ALM-OTRn.LOG (n: 1 to 5)
Saved all history		ALLLOGn.LOG (n: 1 to 3)
Field network configuration file	/FNET	
Ethernet/IP configuration file		EIPCONFDATA.BIN

■Data file of extended common variable

Extended common variables III (#90000 to #97999) are not included in the common variable data file "COMMON.VAR" and the extended common variable data file "EXCOMMON.VAR". Data file of extended common variables III is saved with the fixed file name in the directory in an SD card as follows. (Select the SD card by pressing [DS] for the device on the I/O screen.)

It can be input/output in binary format.

Data type	SD card directory path	Fixed file name
Extended common variables III	EXCOMVAR	COMVARn.DAT (n: 1 to 1000)

12.8.3 Edit Lock B and C

This function prohibits editing, erasing, etc., of the machining programs B and C, and protects the machining programs in the device [Memory].

Machining program		Edit Lock B	Edit Lock C
A	1 to 7999 10000 to 99999999	---	---
B	(user standard program) 8000 to 8999	Editing prohibited	---
C	(MTB custom) 9000 to 9999	Editing prohibited	Editing prohibited

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	○	×	×	○	○	×
Input/Output	Transfer	○	×	×	○	○	×
	Compare	○	×	×	○	○	×
	Copy	○	×	×	○	○	×
	Merge	○	×	×	○	○	×
	Rename	○	×	×	○	○	×
	Erase	○	×	×	○	○	×
Monitor	Buffer correction	○	×	×	○	○	×

Note

(1) When the multiple part system specification is valid, the machining programs of all the part systems are protected by edit lock B and C.

(2) When multiple serial files or a wildcard (*) is designated for input/output function, programs except subject to edit lock will be processed.

(3) When a multiple programs are input to the memory without designating the file name like serial input or program batch input, the message "Edit lock B" or "Edit lock C" appears and the input is interrupted if the program which is subject to edit lock is detected.

(4) A machining program in the USB memory is assumed as machining program A whatever its file name is, it is not subject to the protection of Edit Lock B or C.

12.8.4 Data Protection Keys

Data protection key prevents various data from being set or erased.
The key name differs according to the MTB. Refer to the instruction manual issued by the MTB for details.

The following data can be protected with the data protection key:

- All tool data and the coordinate system preset data using the origin set
- User parameters and common variables
- Machining programs
- Files in the memory card
- Files in the DS

The data protection key is enabled to prohibit the setting, erasing, etc. while the PLC signal is OFF (B contact).
Data protection key has higher protection level than that of Data protection by user's level.

12.8.5 The Batch Input/Output the Machining Program of NC Memory

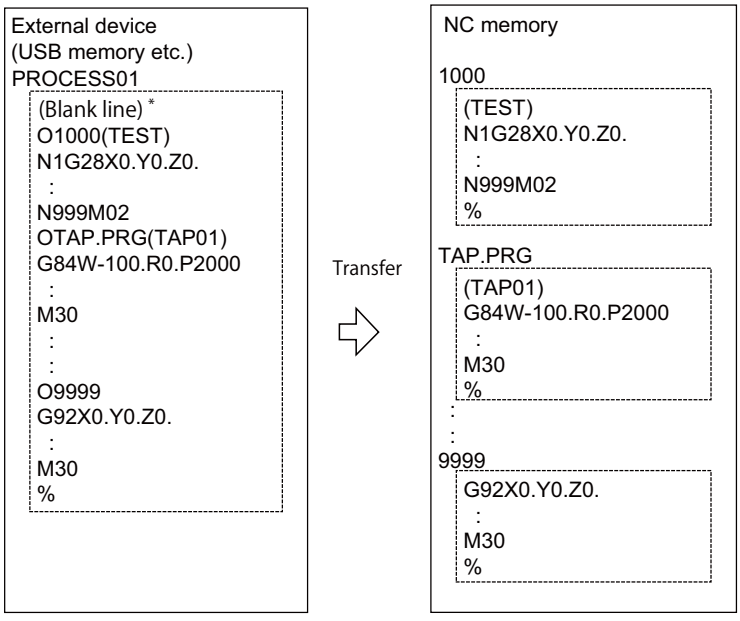
Mainte – I/O

A single file containing two or more machining programs can be transferred to NC memory or NC memory 2 by dividing the file. Machining programs united in one file on NC memory or NC memory 2 can be transferred to an external device.

The targets are the machining programs of the user.

Batch input and verify to NC memory or NC memory 2

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 [Area change] to select file setting column A.

(2) Designate the transfer source device and directory.

(3) Designate the file which consists of two or more machining programs.

(4) Press the menu [Area change] to select file setting column B.

(5) Designate the transfer direction device [Memory].

(6) Press the menu [Dir] - [Program all].

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 "TEST-CUT" is designated as the transfer direction file name, the top program is transferred as not "1000" but "TESTCUT".

(7) Press [Trnsfr A->B].

(8) Press the [Y] or [INPUT] key.

When the same machining program already exists in transfer direction device, the operation message "Overwrite this file?(Y/N)" appears.
The transferring the following program is continued by selecting [Y] or [N] key.

(9) Press [Compare A:B].

Note

(1) When programs are input in a batch to NC memory or NC memory 2, always select [Program all] for NC memory or NC memory 2 as transfer destination. If [Program all] is not selected, one file is transferred (No batch).

(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 "12.8.3 Edit Lock B and C".)

(3) When the protection of the machining program is valid, the program cannot be transferred/verified. (Refer to "12.8.4 Data Protection Keys".)

(4) The program during automatic operation, program restart or program checking cannot be overwritten. The operation message "Executing automatic operation", "Program restarting" or "Program checking" appears, and transfer is interrupted.

(5) If the transfer destination includes the file being edited on the custom screen, the operation message "Can't write into being edited file" appears and the transfer may be interrupted.

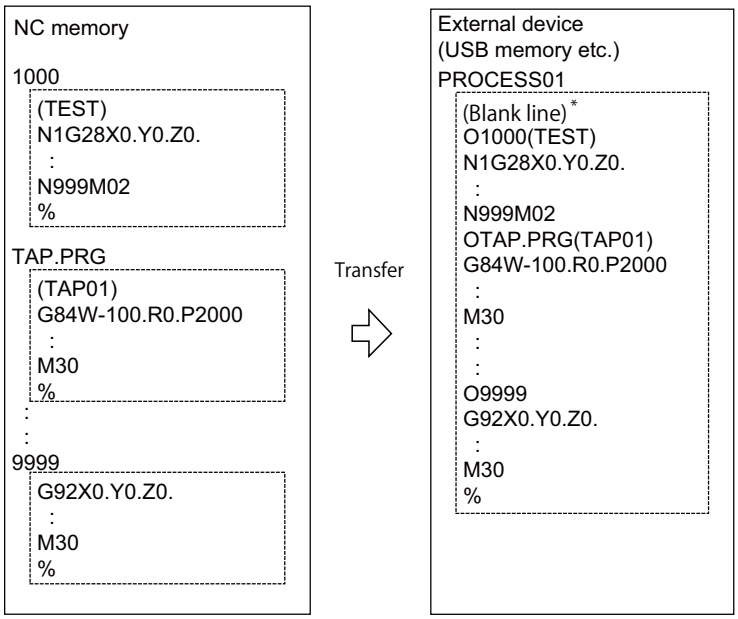
(6) The first line of the transfer source file is ignored.

(7) 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 all	Program
File name	(Null or file designation)	(Null or file designation)

Batch output and verify from NC memory or NC memory 2

Example of "NC memory -> the external device" of machining program batch transfer



When the external device is serial, the top line is %.

(1) Press [Area change] to select file setting column A.

(2) Designate the transfer source device [Memory].

(3) Press the menu [Dir] - [Program all].
Do not specify the file name. If the name is specified, an error will occur.

(4) Press [Area change] to select file setting column B.

(5) Designate the transfer direction device and directory.

(6) Designate the transfer direction file to input multiple machining programs.
When the file name is not designated, output to the file "ALL.PRG" of the designated directory.

(7) Press [Trnsfr A->B].

(8) Press the [Y] or [INPUT] key.
When the same machining program already exists in transfer direction device, the operation message "Overwrite this file?(Y/N)" appears.
The transferring the following program is continued by selecting [Y] or [N] key.

(9) Press [Compare A:B].

Note

(1) When programs are output in a batch from NC memory or NC memory 2, always select [Program all] for NC memory or NC memory 2 as transfer source. If [Program all] is not selected, the program is transferred by one program per one file.

(2) Any programs in NC memory or NC memory 2 with edit lock enabled will not be transferred. All other files will be transferred. The verification is executed excluding the programs which are subject to edit lock. (Refer to "12.8.3 Edit Lock B and C".)

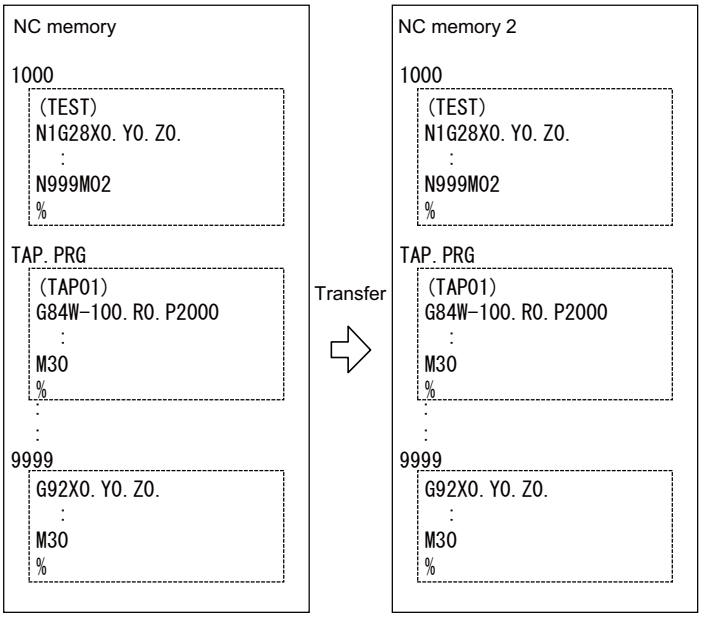
(3) When the protection of the machining program is valid, the program cannot be transferred/verified. (Refer to "12.8.4 Data Protection Keys".)

(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 all	Program
File name	(Null)	* (Wildcard)

Batch output and verify between NC memory and NC memory 2

Example batch transfer of machining programs from NC memory to NC memory 2



(1) Press [Area change] to select file setting column A.

(2) Designate the transfer source device [Memory].

(3) Press the menu [Dir] - [Program all].
Do not specify the file name. If the name is specified, an error will occur.

(4) Press [Area change] to select file setting column B.

(5) Designate the destination device [Memory2].

(6) Press [Dir] - [Program all].
Do not specify the file name. If one is specified, an error will appear.

(7) Press [Trnsfr A->B].

(8) Press the [Y] or [INPUT] key.
When the same machining program already exists in transfer direction device, the operation message "Overwrite this file?(Y/N)" appears.
The transferring the following program is continued by selecting [Y] or [N] key.

(9) Press [Compare A:B].

Note

(1) Similarly, a batch input can be performed with NC memory 2 as device A and NC memory as device B.

(2) [Program all] should be set for the directories for both devices A and B. If [Trnsfr A->B] is pressed with [Program] set for one of the directories, no transfer will occur and the operation message "Setting error" will be displayed.

(3) 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 "12.8.3 Edit Lock B and C".)

(4) When the protection of the machining program is valid, the program cannot be transferred/verified. (Refer to "12.8.4 Data Protection Keys".)

(5) The program during automatic operation, program restart or program checking cannot be overwritten. The operation message "Executing automatic operation", "Program restarting" or "Program checking" appears, and transfer is interrupted.

(6) If the transfer destination includes the file being edited on the custom screen, the operation message "Can't write into being edited file" appears and the transfer may be interrupted.

(7) Set the parameter "#1166 fixpro" (fixed cycle editing) to "0". If a value other than "0" is specified, the operation message "Setting error" is displayed.

Precautions

(1) When the machining program batch input/output function of NC memory or NC memory 2 is used, the machining program name cannot contain "()". 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 [Program all] is selected for NC memory or NC memory 2, no files can be deleted, renamed, or merged. When the files are deleted in a batch, select [Program] for directory, and a wildcard (*) for file name.

(4) For other precautions, refer to "12.2 Transferring a File".

12.8.6 Format Memory Card/DS/USB Memory

This section explains how to format the memory card (front-side SD card), DS (built-in SD card) and USB memory.

(Example) Formatting memory card

(1) Press the menu [MemCrd format]. → The message confirming the format "OK?(Y/N)" appears.

(2) Press the [Y] or [INPUT] key. → The memory card is formatted.
During format, the message "Executing format" appears and the message "Format complete" appears when the format is completed.

When formatting DS or USB memory, press the menu [DS format] or [USB Memformat] in the procedure (1).

Note

(1) When the memory card, DS and USB card are less than 2GB, the format is executed with FAT16, and when they are 2GB to 32GB, format is executed with FAT32.
When formatting the memory card, DS or USB memory which exceeds 32GB, the message "Format error" appears and the format cannot be executed.

(2) The format cannot be executed during automatic operation.

12.9 Performing Batch Backups of NC Memory Data

Mainte – 

The all backup screen of the maintenance (Mainte) screen is used to perform batch backups of the NC memory data to an external device. Also, the data can be restored from the external device to the NC memory in a batch.

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.
This operation is protected according to the operation level.

The following devices can be designated to store the all-backed up data.

HD	Hard disk (This cannot be selected on M800VS/M80V Series.)
Memory card	Front-side SD card
DS	[M800VS/M80V Series] Back-side SD card of the display unit [M800VW/M80VW Series] Control unit-side SD card
USB memory	-

Files to be backed up/restored are as follows:

Screen display	File name	Data type
System data	SRAM.BIN	SRAM data (binary files) Programs, parameters, R registers, etc.
	ALL.PRM (*1)	Parameters
	ASSEMBLY.INF (*1)	System configuration
	NCMEMORY2.IMG (*3)	Machining program data of NC memory 2
	COMVARn.DAT (*3)(*4)	Extended common variable data (for extended common variable III)
	ALL.PRG/ALL2.PRG (*1)(*3)(*7)	Program batch file of NC memory/NC memory 2
	TOOL.OFS (*1)(*3)(*7)	Tool compensation amount data
	COMMON.VAR (*1)(*3)(*7)	Common variable data
	EXCOMMON.VAR (*1)(*3)(*7)	Extended common variable data (for extended common variable I/II)
	WORK.OFS (*1)(*3)(*7)	Workpiece offset data
	TOOLALL.DAT (*1)(*3)(*7)	All tool data
	MMACRO.DAT (*3)(*8)	Machine tool builder macro file
	MMACVAREX.DAT (*3)	Machine tool builder macro variables extended file
Ladder	USERPLC.LAD	User PLC program
	EIPCONFDATA.BIN	EtherNet/IP configuration file
	PLCMSG_XXX.TXT (*2)	PLC message file for MTB
	PLCUMSG_XXX.TXT (*2)	PLC message file for users
	MCRMSG_XXX.TXT (*2)	Macro alarm message file
Safety parameters	SAFEPARA.BIN	Safety parameter (binary file)
Safety ladder 1	SAFEPLC1.LAD	Safety PLC program 1
Safety ladder 2	SAFEPLC2.LAD	Safety PLC program 2
APLC data	APLC.BIN (*10)	C language module created by user
Custom data	-	Custom screen data Motion control release file (*5)(*11) HPU custom module file (*5)(*6)

Screen display	File name	Data type
NCAID data	-	[M800VW/M80VW Series] NCAID data (*3)(*9) (NCAID application, setting file, database, etc.)

(*1) These data can be backed up only. It cannot be restored.

(*2) "XXX" indicates the abbreviation for each language.

(*3) This data is not included in the master data.
Master data is used to back up or restore the default data set by the MTB. The menus [Master Backup] and [Master Restore] are used for this operation.
Master data is always backed up in internal memory regardless of the selected device.

(*4) "n" indicates a value in the range from 1 to 1000.

(*5) The data can be backed up only when the MTB password has been input.

(*6) The data is backed up only when the parameter "#1762 cfgPR12/bit0" (NC-HPU optical communication: communication function valid) is set to "1".

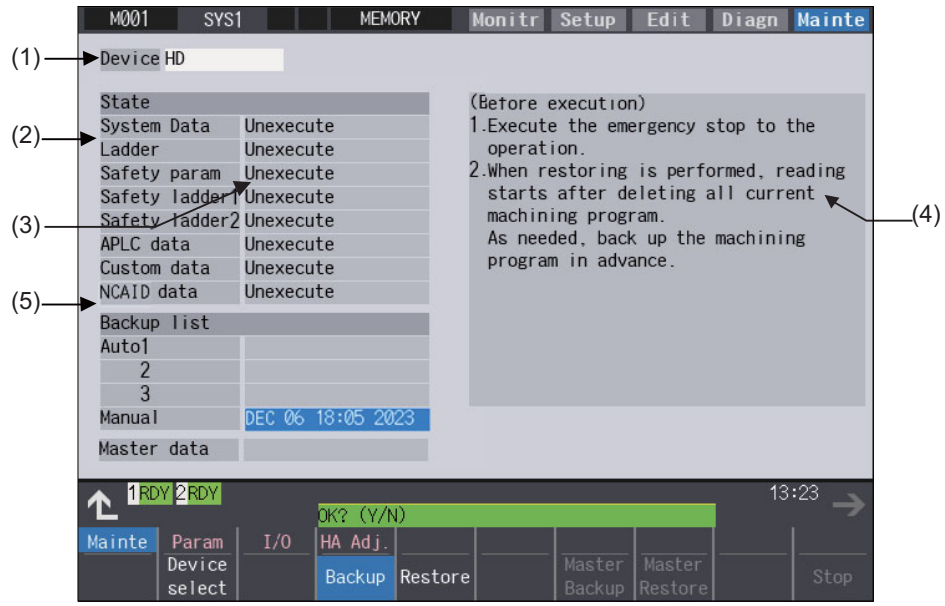
(*7) The data is backed up only when the parameter "#1759 cfgPR09/bit3" (Individual NC data backup valid) is set to "1" and manual backup is performed.

(*8) The machine tool builder macro program is included in this file when a ROM area is selected as the storage area of the machine tool builder macro program (the parameter "#1049 mmac_R" is "1"). This file is backed up/restored only when the "machine tool builder macro memory capacity (512 KB/800 programs)" or "machine tool builder macro memory capacity (1024 KB/1000 programs)" specification is valid. When a SRAM area is selected (the parameter "#1049 mmac_R" is "0"), the machine tool builder macro program is included in SRAM.BIN, and this file is not used.

(*9) The data is backed up/restored only when NCAID is installed, the parameter "#19250 NCAID con. valid" is set to "1", and "#1725 cfg25/bit3" (NCAID backup selection) is set to "0".

(*10) The data is backed up/restored only when the APLC release specification is valid.

(*11) The data is backed up/restored only when the motion control release specification is valid.



Display items

<table>
<tr><th colspan="2">Display items</th><th>Details</th></tr>
<tr><td>(1)</td><td>Device name</td><td>This displays the selected device name.</td></tr>
<tr><td>(2)</td><td>File name</td><td>This displays the file name being backed up/restored.
The following items are displayed starting from the top.
"System Data", "Ladder", "Safety param", "Safety ladder1", "Safety ladder2", "APLC data", "Custom Data" and "NCAID data"</td></tr>
<tr><td>(3)</td><td>Execution status</td><td>This displays the status being executed.
Processing is executed in the order of "System Data", "Ladder", "Safety param", "Safety ladder1", "Safety ladder2", "APLC data" and "NCAID data". (*1)</td></tr>
<tr><td>(4)</td><td>Warning message</td><td>This displays messages at the start and end of backup/restore processing.</td></tr>
<tr><td>(5)</td><td>Backup list</td><td>This displays the backup date list.
This date is the time stamp of system data .
<table>
<tr><th>Backup list</th><th>Explanation</th><th>Backup format</th></tr>
<tr><td>Auto 1
Auto 2
Auto 3</td><td>Data that was automatically backed up.
Display in the backup date descending order. The latest 3 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
(internal memory)</td><td>Data that was backed up on this screen.
Generally, this is the factory settings data.</td><td>Manual</td></tr>
</table>
The above data can be selected at restore processing. (*2)</td></tr>
</table>

(*1) APLC data cannot be backed up/restored if the APLC release specification is invalid.

(*2) "Auto 1 to 3" data display when the device set by the parameter "#8919 Auto backup device" is selected.

Menus

Menus	Details
Device select	This displays the sub-menu for "Device" selection.
Backup	This executes backup processing.
Restore	This executes restore processing.
Master Backup	This backs up NC data to the built-in memory.
Master Restore	This restores NC data to the built-in memory.
Stop	This stops processing.

12.9.1 All Backup

This section explains the procedures to back up NC data in a memory card in a batch.

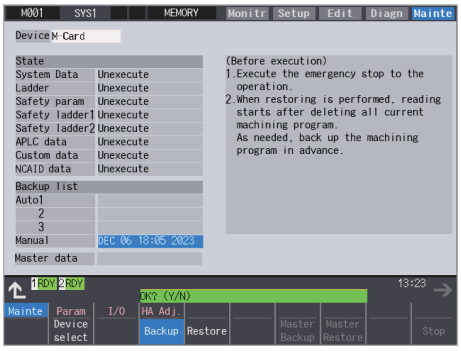
Operation method

(1) Press the main menu [Mainte].

(2) Press the menu [All backup]. → If password input is required, input the MTB password.

(3) Press the menu [Device select], and select the menu [Memory card] as a device to save the data. → Make sure the device "Memory card" is displayed.

(4) Press the menu [Backup]. → The confirmation message appears.

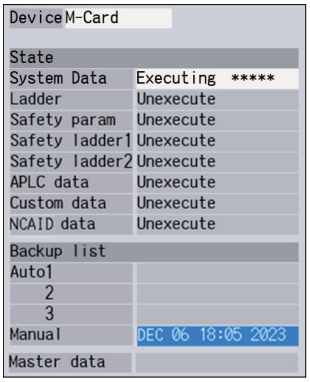


<Note>

- Press [N] or to cancel this process.

(5) Press the [Y], [y] or [INPUT] key. → The backup starts.

"Executing" and "*" are displayed for the state of file under processing.



<Note>

- If any NC data file exists in the backup folder, the confirmation message "Overwrite this file?(Y/N)" appears. In this case, press the [Y], [y] or [INPUT] key to start backup.
- If the tool data is being sorted, the operation message "File access error" is displayed and the backup cannot be executed.
- For the custom data, the backup cannot be canceled during its execution.

(6) The backup is completed without error. → The states of backup-complete files turn to "Complete", the date displayed in "Backup list" will be updated.

<Note>

- If the APLC release specification is invalid, the APLC data cannot be backed up and "Unexecute" is displayed.

Note

(1) If an error occurs during the backup execution, the display is as follows:

- Operation message: "File access error"
- Execution state: "Error" for the file on which an error has occurred
- Selected area: Clear

(2) Even when the data protection key (for memory card or SD) is valid, the data can be backed up in the memory card or DS.

(3) On the Windows-based display unit, if the size (the size in the actual memory) of custom data exceeds 48 MB, the operation message "Skipped custom data due to over capacity" appears and the custom data cannot be backed up. Note that the size in the actual memory and the size on Windows may differ.

12.9.2 All Restoration

The following explains the procedure to restore the various NC data backed up to memory card in a batch.
The files which has been automatically backed up can be restored only when the device set by the parameter "#8919 Auto backup device" is selected.

Note

(1) The presence/absence of each file for the transfer destination; system data, ladder, APLC data, and custom data (for only M80V Series), is not checked when the restoration is executed.

(2) Each file for the system data, ladder, APLC data, custom data and NCAID data are rewritten when the restoration is executed.

(3) In the following cases, an error message appears and the restoration is not executed.

- During the automatic operation
- One of the data protection key 1, 2 and 3 is valid.
- The NC serial number does not match the manufacturing number of the backup data.
- The "System data" file does not exist in the transfer source.
- The ladder larger than 256 KB is stored and all backup is executed when the large-capacity PLC specification is valid, and then the specification is switched to invalid.

Operation method

(1) Press the main menu [Mainte].

(2) When the smart safety observation is valid, press the menu [Psswd input]. → Input the safety password.

<Note>
- If the safety password is not entered, the safety parameters and the safety ladders cannot be restored.

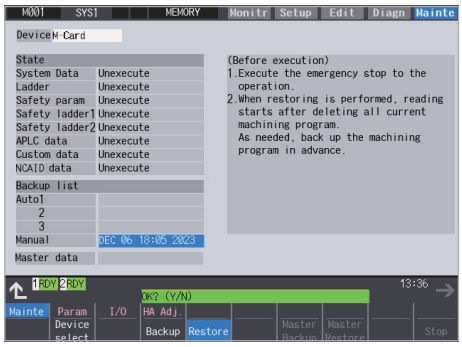
(3) When the Windows security function is valid, click the menu [Scty. func] - [Scty. update].

<Note>
- If the menu [Scty. update] is OFF, the restored NCAID application does not work.

(4) Press the menu [All backup]. → If password input is required, input the MTB password.

(5) Press the menu [Device select], and select the menu [Memory card] as a device where the data to be restored is stored. → Make sure the device "Memory card" is displayed.

(6) Press the menu [Restore]. → The confirmation message appears.



<Note>

- Press [N] or to cancel this process.
- If the target device set for auto backup does not match the currently selected device, "Manual" is highlighted as the initial display position of the cursor.

(7) Move the cursor to select the file. Press the [INPUT] key.

The cursor moves as follows:

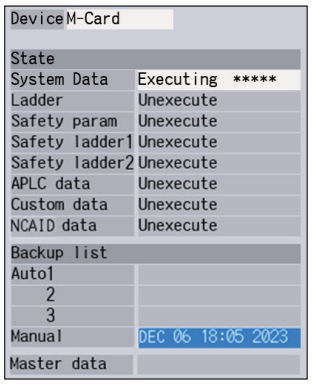
[↓] key: Auto1 -> Auto2 -> Auto3 -> Manual -> Auto1 ...

[↑] key: Auto1 -> Manual -> Auto3 -> Auto2 -> Auto1 -> …

→ If there are no files at the position where the cursor moved to, the operation message "Select directory to restore" appears. Select the file again and then press the [INPUT] key.

(8) Press the [Y], [y] or [INPUT] key. → The restoration starts.

"Executing" and "*" are displayed for the state of file under processing.



<Note>

- If PLC is in operation, the message "PLC running. Does it stop?(Y/N)" appears.
  Press the [Y], [y] or [INPUT] key to continue the restoration.
- If the tool data is being sorted, the operation message "File access error" is displayed and the restoration cannot be executed.
- For the custom data, the restoration cannot be canceled during its execution.

(9) The restoration is completed without error. → The states of restoration-complete files turn to "Complete", the operation message "Restore complete" appears.
"Unexecute" is displayed for the file which could not be restored, for instance, due to the nonexistence of the file.

Note

(1) When the restoration fails, perform the restoration again otherwise it may cause an abnormal operation.

(2) Restored data writes over the previous absolute position data. Carry out the absolute position detection again after restoration.

(3) If you attempt to restore custom data while custom data has already been installed, the Windows-based display unit shows the operation message "File access error", which may disable restoration. Use the integrated installer to delete the data from the NC before restoring custom data.

13

Daily Maintenance

Some parts will not function in a hardware manner when the life is reached, so these should be replaced before the life is reached.
For details of the inspection and maintenance of the drive section, refer to the instruction manuals for each drive unit.

Class	Name	Life	Inspection/replacement
Daily Maintenance	Touchscreen		When needed (when poor sensitive or dirty)
	Escutcheon (display unit area, keyboard etc.)		Once/two months (Accordingly when dirty)
Periodic Maintenance	Battery (lithium battery)	Cumulative data holding time: 45,000 hrs	When battery voltage drop caution alarm occurs (Guideline: approx. 5 years)

Note

(1) Replace the battery within one month after the alarm "Z52 Battery fault 0001" appears on the NC screen.
(2) The internal data may have been damaged if the alarm "Z52 Battery fault 0003" appears.

Touchscreen

⚠ CAUTION

Always keep the touchscreen surface clean. When it gets dirty, remove the dirt immediately. Do not perform any operation on the dirty touchscreen.

(1) Cleaning the touchscreen
- When metal powder or sand dust is on the touchscreen, remove it to prevent the scratches.
- Wipe the screen gently with a soft and clean cloth.

(2) Precautions for use
- The touchscreen may become poor sensitive, or even may not react when it gets dirty.
- Operate the touchscreen with fingers. The touchscreen does not react with nails.
- The use of gloves, work gloves, etc. will have effects such as the deterioration of the operation feeling depending on the material and thickness of the gloves.
- The polarizing plate (display surface) of the touchscreen surface can be easily scratched, so be careful during handling.
- Glass is used in the touchscreen. Be careful not to drop the touchscreen 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.
- Electronic parts are used in the touchscreen, so be careful of static electricity when handling.
- Never disassemble the Touchscreen. Doing so will damage the panel.

(3) Precautions for storage
- Do not store the card in locations having a high temperature or humidity. (Store within the storage temperature range.)
- When storing the touchscreen as an individual unit, be sure that other objects do not touch or hit the polarizing plate (display surface).
- When storing the touchscreen for long periods, be sure to store in a dark place away from exposure to direct sunlight or fluorescent light.

Cleaning the escutcheon

(1) Prepare the rear side of the escutcheon to clean.

(2) 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.

14

Appendix 1: Rotation Center Error Measurement Function (M System) / Rotation Center Error Compensation Function

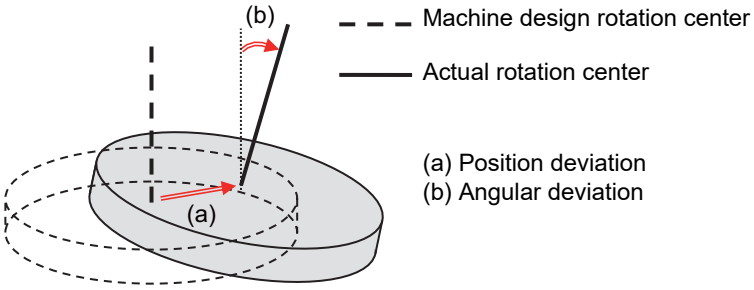
Mainte – Mainte

In the machine having a rotary axis, the rotation center deviates due to the assembly error of the rotary axis. Thus, if a machining is carried out with the rotary axis, the machining accuracy is impaired because an error is caused in the relative position between the tool and workpiece.

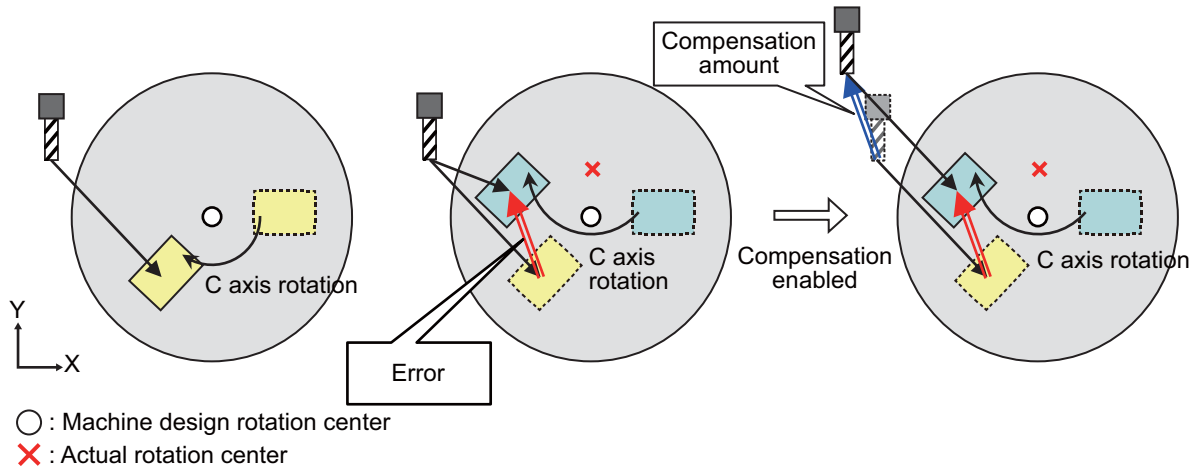
There are two types of rotation center deviation. One is "position deviation", in which the rotation center is shifted in the planar direction, and the other is "angle deviation", in which the rotation center inclines.

Higher-accuracy machining is realized by compensating for these two types of deviation.

[Position deviation and angular deviation]



[Compensation for the deviation of the rotation center]



[No deviation]	[With deviation] The error arises between tool and workpiece; accuracy decreases.	[When this function is enabled] The error between the tool and workpiece is compensated for by using the measured deviation (parameter).

14.1 Rotation Center Error Measurement Function (M System)

This function executes automatic measurement with the touch probe on the reference sphere to calculate the rotation center error compensation amount.
A reference sphere and a touch probe which can input skip signals to the CNC are required to use this function. The movement of the touch probe, the rotation of the axis, the reading of the coordinate, and the calculation of the deviation can automatically be performed with this function. And calculated values of the deviation are to be set to the the rotary axis configuration parameters.

Outline of the measurement

■Precaution for the measurement

The center of the rotary axis is measured for the distance from the reference position of the tool length measurement or workpiece coordinate system offset measurement. Thus, the tool length compensation amount of the the touch probe must be measured under the same condition of the reference position of the tool length measurement or workpiece coordinate system offset measurement. If the reference position on measuring the tool length of the touch probe differs from that of the said measurements, the center of the rotary axis cannot be measured correctly.

■Items to be prepared

- Reference sphere
- Touch probe which can input skip signals to the CNC

For details of usable touch probe, consult the MTB.

■Measurement procedure

Set a reference sphere.



Install a touch probe and confirm the skip signal (X6F8) is input when it makes a contact.



Input the rotation center error measurement macro program in the NC memory.



Set the measurement conditions.



Start the measurement.

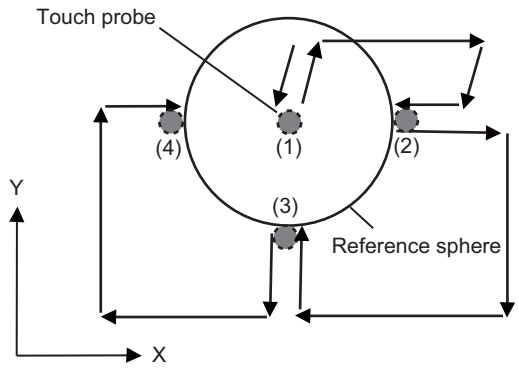


Write the measurement results in the parameters.

■Movement of the measurement

Once the measurement starts, four locations (1) to (4) as shown in the following figure are automatically calculated for 5 types of axes' angle combination in the table.
The 1st rotary axis refers to A axis or B axis, and the 2nd rotary axis refers to C axis.

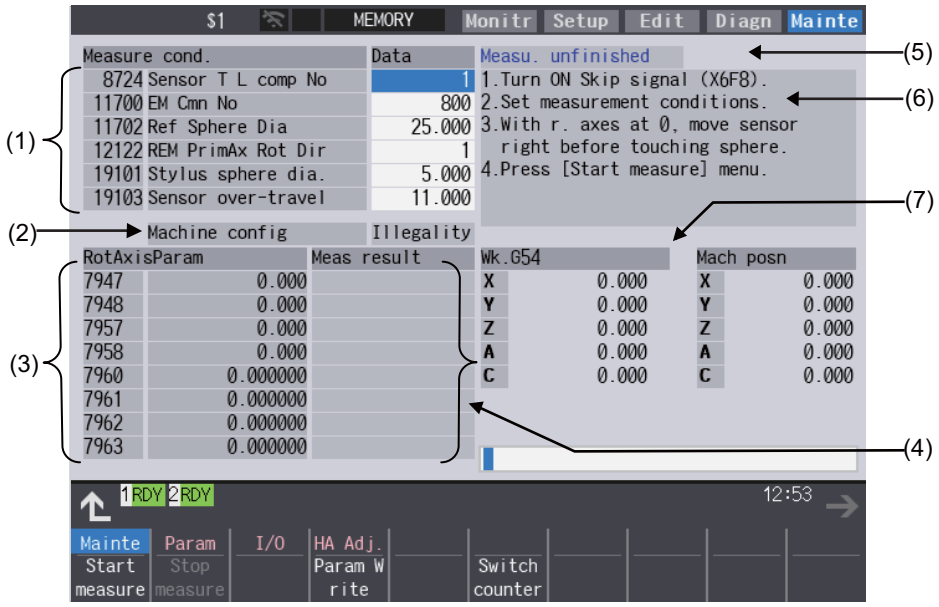
	1st rotary axis	2nd rotary axis
1	0°	0°
2	0°	90°
3	0°	180°
4	±45° (*1)	0°
5	±90° (*1)	0°



(*1) The rotation direction of the 1st rotary axis can be set in the measurement condition (#12122) on the rotation center error measurement screen.

14.1.1 Screen Configuration

MTB password is required for the operation on the rotation center error measurement screen.



Display items

Display items			Details	
(1)	Measurement condition parameters		Conditions to perform the rotation center error measurement can be set in these parameters.	Setting range
	8724	Sensor T L comp No	This sets the tool length compensation No. of the touch probe to be used for the measurement.	1 to 999
	11700	EM Cmn No	This sets the starting number of the common variables to be used for the measurement. Consecutive 8 common variables from the set number are used for the measurement.	Common variables The value of "set number + 7" must be within the range of the common variables.
	11702	Ref Sphere Dia	This sets the diameter of the reference sphere to be used for the measurement.	10 to 100 (mm)
	12122	REM PrimAx Rot Dir	This selects the rotation direction of the primary axis (A axis or B axis) when the measurement is performed. Select the direction allowing for the stroke range of your machine.	0: Direction of 0˚ to 90˚ 1: Direction of 0˚ to -90
	19101	Stylus sphere dia.	This sets the diameter of the stylus sphere to be used for the measurement.	1 to 10 (mm)
	19103	Sensor overtravel	This sets the overtravel distance (overtravel allowance amount and over stroke) of the touch probe to be used for the measurement. Set the smallest overtravel distance among all the axes.	0 to 100 (mm)
(2)	Machine configuration		This displays the machine configuration when the measurement is performed. The display of this field varies depending on the rotary axis configuration parameters. You cannot set this item.	
(3)	Rotary axis configuration parameters		These are the parameters to be set the deviation obtained by the measurement result.	
(4)	Measurement result		This displays the deviation obtained by the measurement result.	
(5)	Measurement status display		This displays measurement status such as "Measurement ready" or "On mea", etc.	
(6)	Operation guidance		This displays the guidance for the measurement.	

Display items	Details
(7) Counter display	The workpiece coordinates position is displayed on the left side. The machine coordinates position or remaining distance is displayed on the right side. The counter to be displayed can be switched by pressing the menu [Switch counter]. Up to six axes can be displayed on each counter.

Menus

Menus	Details
Start measure	The measurement is ready. While "Measurement ready" or "On mea" is displayed, all the menus become disabled. While "Measure finished" or "Measure canceled" is displayed, they become selectable. Press the menu [Stop measure] to cancel the "Measurement ready" status.
Stop measure	This interrupts the measurement setting. Menus become selectable. While the measurement status "On mea" is displayed, reset the NC first and then press the menu [Stop measure].
Param write	This writes the measurement result to the rotary axis configuration parameter.
Switch counter	This switches the counter type on the right side. "Mach posn" and "Remain distance" are toggled each time the menu is pressed.

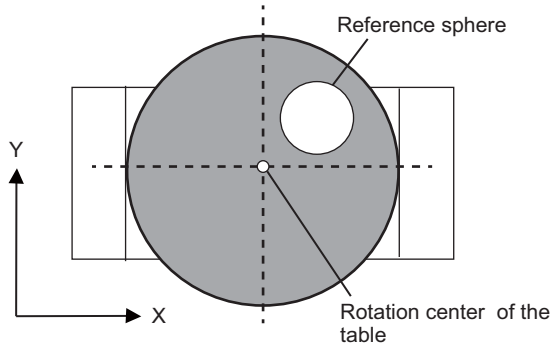
Parameter setting examples

Parameter No.	Details	Setting value (unit)
#8724	Tool length compensation No. of the touch probe	2
#11700	Starting No. of the common variable to store the rotation center error measurement result	800
#11702	Reference sphere diameter	25 (mm)
#12122	Rotation direction of the primary axis, i.e A axis or B axis (0° to 90° direction)	1
#19101	Stylus sphere diameter	5 (mm)
#19103	Overtravel distance of the touch probe	11 (mm)

14.1.2 Measuring the Rotation Center Error Data

Measuring the rotation center error data

(1) Set a reference sphere at the position apart form the rotation center of the table as shown in the figure below.



(2) Install a touch probe.

Confirm the skip signal (X6F8) is input when the touch probe makes a contact before the measurement is performed.

<Note>

- If the measurement is performed in the state that the skip signal cannot be input, the touch probe may be damaged. You can confirm the skip signal input on [Dagn] - [I/F dia] screen. For details, refer to "11.4 Advanced Diagnosis".

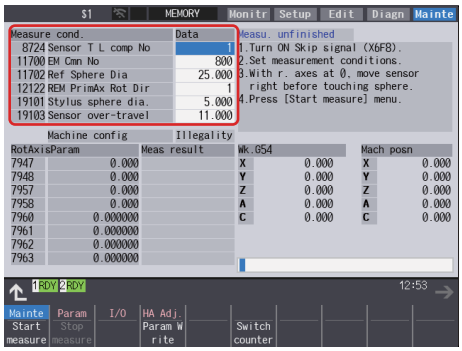
(3) Display [Mainte] - [Param] screen, and set the program No. to be assigned to measurement macro in the parameter "#12121 REM Mcr No".

(4) Display [Mainte] - [Mainte] screen, and input the measurement macro program in the NC memory by pressing the menu [Write RCE mcr].

(5) Press the menu [RCE mea].

(6) Select the target part system where the measurement is performed, and set the conditions for the measurement.
Refer to "Parameter setting examples" in "14.1.1 Screen Configuration" for details of parameter setting.

Make sure that the measurement conditions are correctly set.



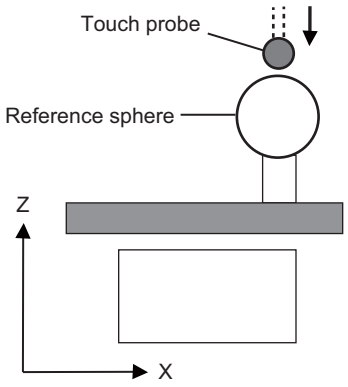
<Note>

- If there is any setting value out of the allowable range, the measurement cannot be started.

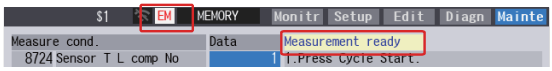
(7) Move the touch probe to the measurement start position by the following procedure.

(a) Set the machine position of each rotary axis to 0°.

(b) With the handle operation, bring the touch probe close to the top of the reference sphere as shown in the figure.



(8) Press the menu [Start measure]. → The measurement status changes to "Measurement ready", and "EM" is displayed on the NC status.



(9) Set the operation mode to MEMORY, and press the CycleStart button. → The measurement starts. The measurement status changes to "On mea".
When the measurement is completed, the measurement status changes to "Measure finished" and the data in "Meas result" are updated.
The "EM" on the NC status disappears.

(10) Press the menu [Write param]. → The message "OK to write measurement results?(Y/N)" appears.

(11) Press the [Y] or [INPUT] key. → The measurement result is reflected to the rotary axis configuration parameters.
The measurement status changes to "Param write done".

Note

(1) In some ladder settings, if the touch probe made a contact with the reference sphere prior to start of the measurement, PLC alarm might be caused.

(2) If NC reset or emergency stop is executed during the measurement, the measurement is interrupted.

(3) The measurement conditions cannot be set after the measurement starts. Operation message "Executing automatic operation" is displayed.

Canceling the rotation center error measurement

Follow the procedure below to cancel the measurement.

(1) While the measurement status "On mea" is displayed, reset the NC first to stop the measurement. → The measurement status changes to "Measurement ready".

(2) Press the menu [Stop measure]. → The confirmation message appears.

(3) Press the [Y] or [INPUT] key. → The measurement is interrupted and the status changes to "Measure cancelled". The "EM" on the NC status disappears.

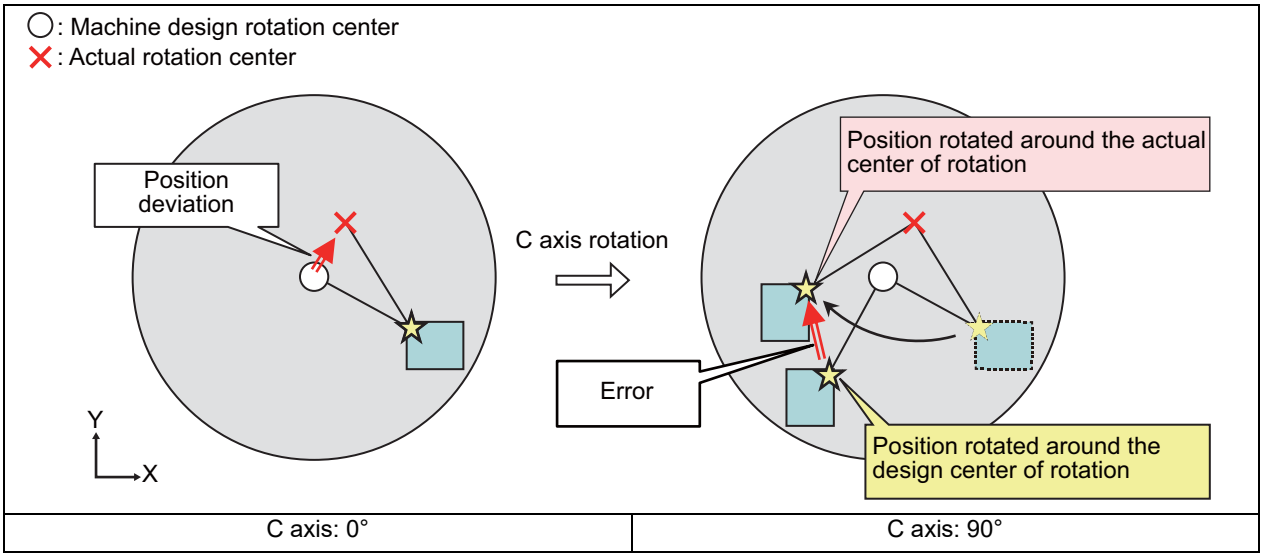
14.2 Rotation Center Error Measurement Function

The position deviation and angular deviation between the actual center of rotation and the center of rotation on the machine design lead to a lower machining accuracy at the cutting point. By setting the position deviation and angular deviation to be compensated beforehand, high machining accuracy is achieved as if these centers of rotation were coincided.

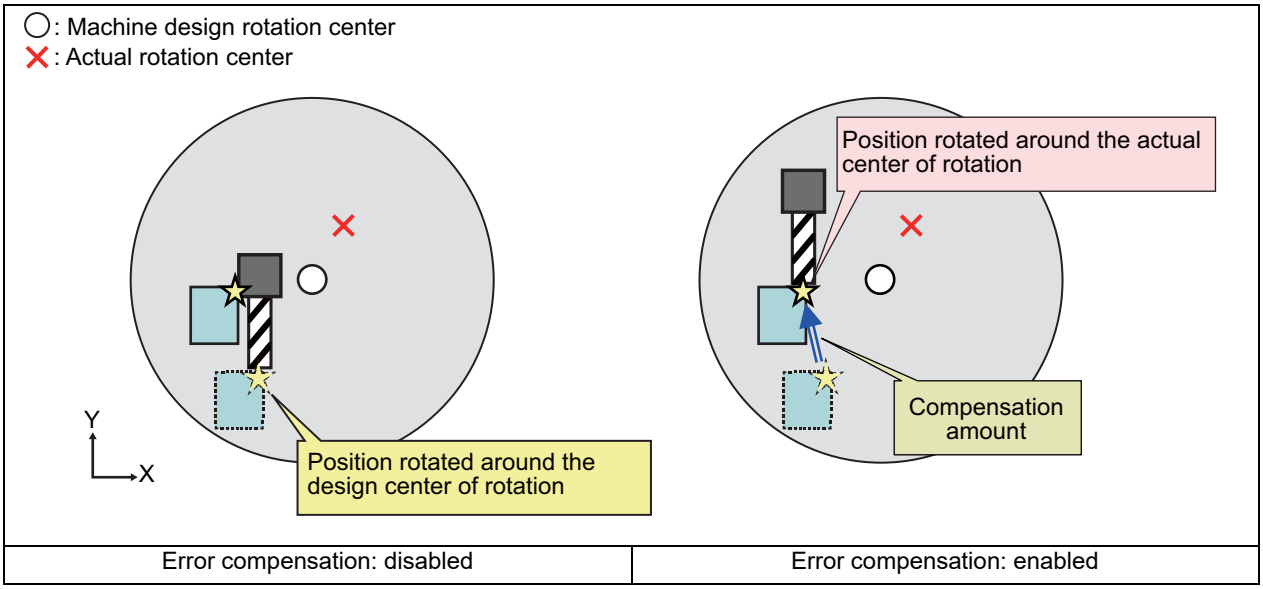
Compensating the position deviation

Due to position deviation, an error arises between the position rotated around the design center of rotation and the position rotated around the actual center of rotation (see the figure in item (a)). This error is compensated for by distributing the compensation amount of this error to the orthogonal axes. This enables cutting to be carried out based on the position rotated at the actual rotation center (see the figure in item (b)).

(a) Error when there is position deviation of C axis



(b) Compensation for position deviation when C axis is 90°



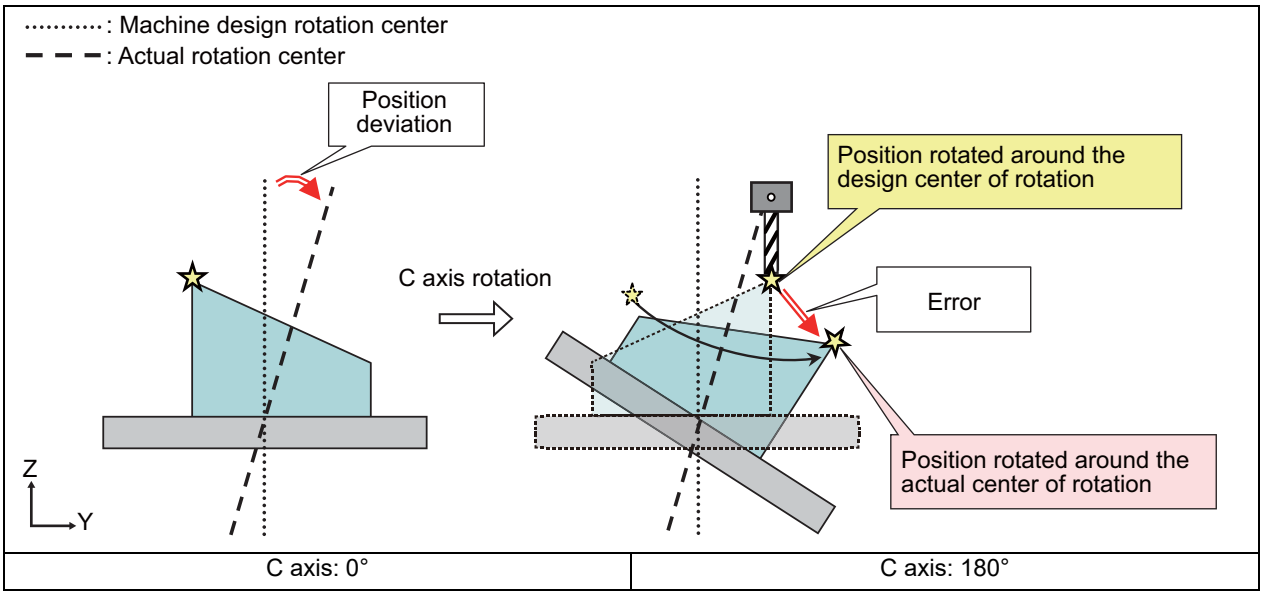
Compensating the angular deviation

Due to angular deviation, an error arises between the position rotated around the machine design center of rotation and the position rotated around the actual center of rotation (see the figure in item (a)). This error is compensated by distributing the compensation amount of this error among the orthogonal axes and rotary axes. This enables cutting to be carried out based on the position rotated at the actual rotation center. Whether compensation amount is distributed to rotary axes or not depends on the MTB specifications (the parameter "#7916 ROT_ERR_MODE" (Rotation center error compensation method)) (see the figure in item (b)).

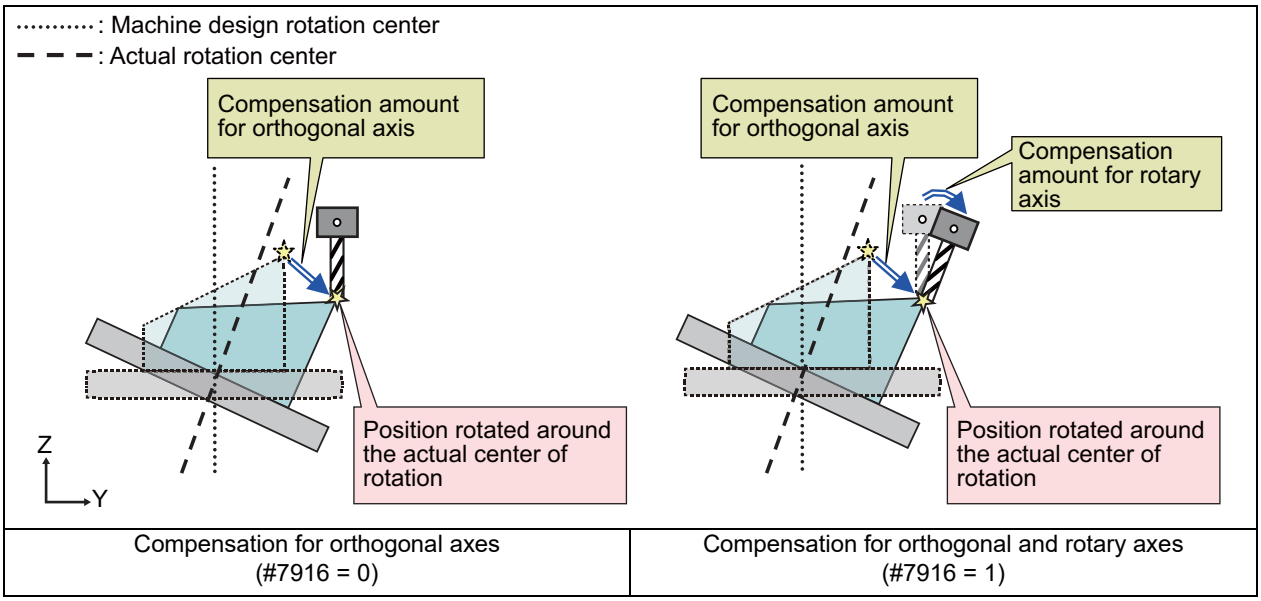
The compensation amount for angular deviation varies depending on the position of the cutting point. By setting the tool length, the tool center point can be corrected. The tool length data to be applied differ between M system and L system (see the table below). If no tool length is set, the compensation amount at the position of the machine end face is applied.

Operation mode	Machine	Tool length data to be applied
Automatic operation	M system	Tool length data specified in tool length offset (G43/G44), tool length compensation along the tool axis (G43.1), or tool center point control (G43.4/G43.5)
	L system	Tool length data specified in T command (tool offset)
Manual operation	M system/L system	The tool length data set in the parameter "#8043 Tool HDL FD OFS"

(a) When there is angular deviation of C axis



(b) Compensation for angular deviation when C axis is 180°



Checking the rotation center error compensation amount

Press the menu [Drv mon] - [Servo unit] on the diagnosis (Diagn) screen to display the rotation/spatial error compensation amount. For details, refer to "11.2.3 Drive Monitor Screen (Servo Unit)".

Relationship with other functions

When the additional functions in the table below are commanded during rotation center error compensation, the operationdepends on the MTB specifications (parameter "#1450 5axis_Spec/bit5").

- "#1450 5axis_Spec/bit5" (0: Machine movement is suppressed. 1: Machine movement is not suppressed.)
- "#1280 ext16/bit1" (Mixed control cancel with reset)
- "#1280 ext16/bit3" (Reset operation for synchronization/superimposition control)

Additional function	"#1450/bit5" setting value	Operation to be performed when the following function is commanded during rotation center error compensation			Angular deviation compensation for superimposed axis or synchronized axis
		Operation of the function on the left		Operation to be performed when the function on the left is commanded	
		At emergency stop	When resetting		
Mixed control (Arbitrary axis exchange control) (G140)	0	Not canceled	Not canceled (*1)	(a) When commanded: Program error (P501) (b) When the axis exchange in the part system is commanded (*2): Operation error (M01 1101)	-
	1	Canceled	Depends on "#1280 ext16/bit1".	Can be commanded. (*3)	-
Mixed control I/II (Cross axis control) (G110)	0	Not canceled	Not canceled (*1)	(a) When the G command issued: Program error (P501) (b) When a command by the PLC signal is issued: Operation error (M01 1035)	-
	1	Canceled	Depends on "#1280 ext16/bit1".	Can be commanded. (*3)	-
Control axis superimposition (G126)	0	Canceled	Not canceled (*1)	(a) When the G command issued: Program error (P520) (b) When a command by the PLC signal is issued: Operation error (M01 1004)	Disabled (*4) Operation error (M01 1003)
	1	Canceled	Depends on "#1280 ext16/bit3".	Can be commanded. (*5)	Enabled (*6)
Arbitrary axis superimposition (G156)	0	Canceled	Not canceled (*1)	Program error (P520)	Disabled (*4) Operation error (M01 1003)
	1	Canceled	Depends on "#1280 ext16/bit3".	Can be commanded. (*5)	Enabled (*6)

Additional function	"#1450/ bit5" setting value	Operation to be performed when the following function is commanded during rotation center error compensation			Angular deviation compensation for superimposed axis or synchronized axis
		Operation of the function on the left		Operation to be performed when the function on the left is commanded	
		At emergency stop	When resetting		
Control axis synchronization between part systems I/II (G125)	0	Canceled	Not canceled (*1)	(a) When the G command issued: Program error (P521) (b) When a command by the PLC signal is issued: Operation error (M01 1037)	Disabled (*4) Operation error (M01 1038)
	1	Canceled	Depends on "#1280 ext16/ bit3".	Can be commanded. (*3)	Enabled (*6)

(*1) This does not depend on the setting of the parameter "#1280 ext16/bit1" or "#1280 ext16/bit3".

(*2) This means a case in which the axis in the part system during rotation center error compensation is designated as the replacement axis.

(*3) The command can be issued, however, machine movement may occur. To ensure safety, issue the command at a 0 degree angle to the rotary axis.

(*4) If angle deviation compensation is commanded to the superimposed axis or synchronized axis, an operation error will occur. This alarm can be reset by turning the "Rotation center error compensation enabled" signal (YD15) OFF.

(*5) The error compensation amount is immediately applied to the superimposed axis or synchronized axis, so machine movement may occur. To ensure safety, issue the command at a 0 degree angle to the rotary axis.

(*6) The angle deviation compensation can be commanded to the superimposed axis or synchronized axis, however, the rotation center error compensation amount of the superimposed axis or synchronized axis is not applied to the reference axis. Therefore, the relative positional relationship between the reference axis and the superimposed axis may be broken. To ensure safety, issue the command at a 0 degree angle to the rotary axis.

Note

(1) If the relative positional relationship between the reference axis and the superimposed axis may be broken or machine movement may occur, issue the command at a 0 degree angle to the rotary axis to ensure safety.

15

Appendix 2: Wireless LAN Function [M800VS/M80V Series]

The wireless LAN functionality enables the NC to exchange data with devices compliant with wireless LAN standards using wireless communication through the built-in wireless module.

To use wireless LAN functionality, set parameter "#75000 WLAN STA/AP mode" to "1" or "2".

The following descriptions are the meanings of the terms used in this chapter.

Term	Definition
Access point (AP)	A wireless access point, which allows wireless stations (e.g. PCs, smartphones) to connect to one another or to another network.
Station (STA)	A wireless client, which communicates with another station directly or through an access point.
Access point (AP) mode	Mode in which the NC unit operates as an access point (AP).
Station (STA) mode	Mode in which the NC unit operates as a station (STA).
Infrastructure mode	Mode in which devices operating as stations connect to the device operating as an access point to communicate with each other.
SSID	The unique identifier of a wireless access point.
ESSID	The unique identifier of a wireless access point and a wireless network. 'SSID' often generically refers to ESSID.
BSSID	The unique identifier of a wireless access point and a wireless network. Access to a specific access point is provided by specifying the MAC address of that access point. BSSID is used when a specific access point needs to be used in roaming, for example.
ANY connection refusal	Mode that blocks access to an access point from station whose SSID are set to "ANY" or unspecified. When ANY connection refusal is enabled for an access point, only stations with the SSID specified can access to it.
AP Isolation	A function that enables an access point to prohibit communications between stations. Sometimes also called the privacy separator function.

Precautions

(1) Ensure that the country/region code that matches the country where the machine is used has been configured correctly. If the configured country/region code does not match the country of operation, the operation may be in violation of the regulatory requirements by radio law. Mitsubishi Electric shall not be held liable for any issues arising from an incorrectly configured country/region code.

(2) If the country/region code needs to be changed, contact the MTB to change it.

(3) Because any wireless device within the area covered by radio wave could access the network in wireless communication, there is risk of content of communications being read by an unintended device. Ensure that appropriate security settings are in place, such as customizing the encryption key that is hard to guess, and restricting devices that can access the network. For details, refer to "Notes on security" on the next page.

(4) If multiple stations are simultaneously connected with the NC unit being used as an access point (AP), the communication speed of wired connections may slow down depending on the communication load.

(5) If communicating device is located on the back side of the machine, the communication might be unstable.

(6) It takes several seconds until the wireless LAN communication becomes available after the screen is displayed.

(7) If an IP address which exists in the same network of wired LAN is set to wireless LAN, the correct communication may not be guaranteed.

(8) "Undo" function cannot be used when setting the parameter "#75007 Encryption key".

(9) Both DHCP function and filtering function of IP filter setting cannot be used simultaneously. When the filtering function of IP filter setting is enabled, the NC unit cannot provide an IP address to DHCP client. Set the filtering function of IP filter setting for wireless LAN disabled.

(10) If any of the following values is set in the parameter "#75052 StartIP DHCPserver", wireless LAN does not work.

- Network address or broadcast address
- Same IP address as that set in "#75001 WLAN IP address"
- IP address for another network

(11) When the NC unit operates as an access point (AP) mode and a DHCP server, the connection may be unstable if a random MAC address is used in a station terminal. Change the setting so that a specific MAC address is used in the station terminal.

Notes on security

One benefit the wireless module offers is that it allows tablets and PCs to exchange data with a wireless access point over radio waves instead of LAN cables, enabling them to access the LAN freely within the area covered by radio waves. On the other hand, because radio waves reach throughout a limited area regardless of obstructions such as walls, if a malicious attacker exploits vulnerabilities, the content of communications could be stolen or malformed packets could be inserted, for example. The following are examples of such problems:

1. The content of communications is snooped on

   A malicious third party could deliberately intercept radio communications and snoop on IDs, passwords and other information.

2. The network gets broken into

   A malicious third party could gain unauthorized access to the network and the following actions could be done.

   - steal personal or confidential information (information leakage)
   - impersonate a valid user and transmit illegitimate information (impersonation)
   - intercept communications and transmit altered information (tampering)
   - destroy data and systems by spreading computer viruses or other harmful programs (vandalism)

You can reduce the likelihood of these problems by configuring the settings of security mechanisms provided by the wireless module and wireless access points.

Ensure that you fully understand the problems that could occur if the product is used without security settings being configured and configure the security settings at your own discretion and risk before using the product.

The following describes how to reduce and prevent these security problems:

(1) When using the NC in the AP mode, ensure the following:

- For the parameter "#75007 Encryption key", set an alphanumeric string that is difficult for others to guess. Avoid consecutive numbers and strings that can be guessed from the MAC address, etc.
- Restrict the IP addresses to allow access using the IP filter setting function.

(2) When using the NC in the STA mode, ensure the following when configuring the wireless LAN router settings:

- For the passphrase used to configure wireless LAN settings, set an alphanumeric string that is difficult for others to guess. Avoid consecutive numbers and strings that can be guessed from the MAC address, etc.
- For the security authentication mode in the wireless LAN settings, use WPA or WPA2.
- To prevent unauthorized access from the Internet, take measures such as disabling PING response to avoid being identified on the Internet.
- Change the password for the login to the management screen to one that is difficult for others to guess. For information about the wireless LAN settings, contact the manufacturer of the wireless LAN router.

(3) When using a device such as a PC or a tablet in the same network, ensure the following:

- Keep the OS, software and anti-virus software updated to the latest version.
- Do not open attached files and hyperlinks from a unreliable or unknown source.

15.1 Configuring the Wireless LAN

The NC unit can be operated in two modes for wireless LAN: access point (AP) mode, where it operates as an access point, and station (STA) mode, where it operates as a station. Note that the NC unit cannot be in both modes at once. Use the parameters to select the mode in which the NC unit will operate.

15.1.1 Configuring for Access Point (AP) Mode

In the access point (AP) mode, the NC unit operates as an wireless access point. Communication with other wireless devices is available in an environment without a wireless router.

Complete the following procedures to configure the NC unit and the wireless stations when the NC unit operates as an access point (AP).

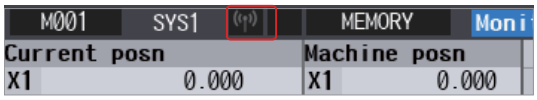
Configuring the NC unit

(1) Set parameter "#75000 WLAN STA/AP mode" to "2".

(2) Configure the following parameters to suit your network environment:

- #75001 WLAN IP address
- #75002 WLAN Subnet mask
- #75005 SSID
- #75006 Encryption mode
- #75007 Encryption key
- #75008 WLAN channel
- #75009 Frequency band

(3) Reboot the NC and verify that the  icon indicating AP mode operation (Not connected to station) is displayed to the left of the NC status.



Configuring wireless LAN stations

Configure the settings of wireless stations by following these steps. The setup procedure varies by wireless station. Refer to the device manual for more information.

(1) Enable the wireless LAN function.

(2) Configure the network settings of the station to suit your network environment. Be sure to set the IP address and subnet mask to be the same network as the NC unit.
When the security mode and encryption method of the wireless LAN need to be set, specify those corresponding to the value of parameter "#75006 Encryption mode".

Verifying connection

Verify the connection by following these steps to confirm that the NC unit and the wireless station have been configured correctly.

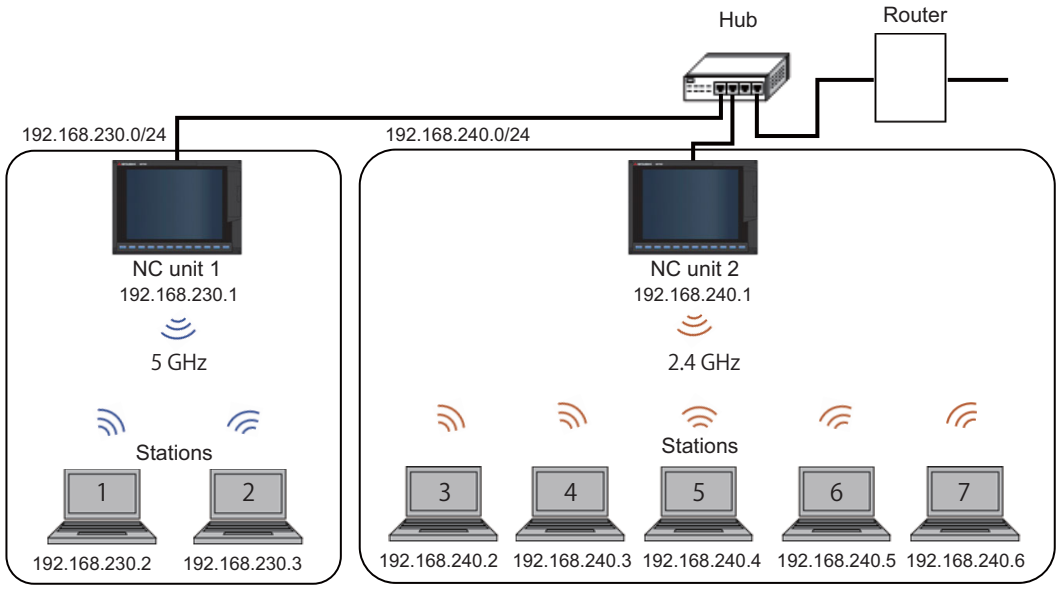
(1) Display the list of available networks (SSID) from the setting screen of the wireless station and select the SSID specified in parameter "#75005 SSID".

(2) When prompted for the encryption key, enter the one specified in parameter "#75007 Encryption key".

(3) Wait until the icon indicating AP mode operation (Connected to station) is displayed to the left of the NC status. Connection has completed when the  icon is displayed.

Example of network configuration

The following shows an example of network configuration when the NC unit operates as an access point (AP).



[Configuration example of wireless LAN parameters]

Parameters		Setting of NC unit 1	Setting of NC unit 2
75000	WLAN STA/AP mode	2	2
75001	WLAN IP address	192.168.230.1	192.168.240.1
75002	WLAN Subnet mask	255.255.255.0	255.255.255.0
75005	SSID	CNC-UNIT1	CNC-UNIT2
75006	Encryption mode	0	1
75007	Encryption key	UNIT1-KEY	UNIT2-KEY
75008	WLAN channel	0	0
75009	Frequency band	1	0

[Configuration example of wireless station]

Item	Setting of Station 1	Setting of Station 3
IP address	192.168.230.2	192.168.240.2
Subnet mask	255.255.255.0	255.255.255.0
Network name (SSID)	CNC-UNIT1	CNC-UNIT2
Security mode	WPA-PSK	WPA2-PSK
Encryption type	TKIP	AES
Encryption key	UNIT1-KEY	UNIT2-KEY

15.1.2 Configuring for Station (STA) Mode

In the station (STA) mode, the NC unit operates as a wireless station. The NC unit is connected as a wirelss station in an environment where the wireless LAN is available.

Complete the following procedures to configure the NC unit and the wireless access point when the NC unit is operated as a station (STA).

Configure the wireless access point

Configure the settings by following the manual or other documentation for the wireless access point (e.g. wireless router).

Configure the NC unit

(1) Set parameter "#75000 WLAN STA/AP mode" to "1".

(2) Configure the following parameters to suit your network environment. Be sure to set the IP address and subnet mask to be the same network as the wireless access point.
Set the SSID and encryption key by specifying the values set for the wireless access point which the NC connects to.

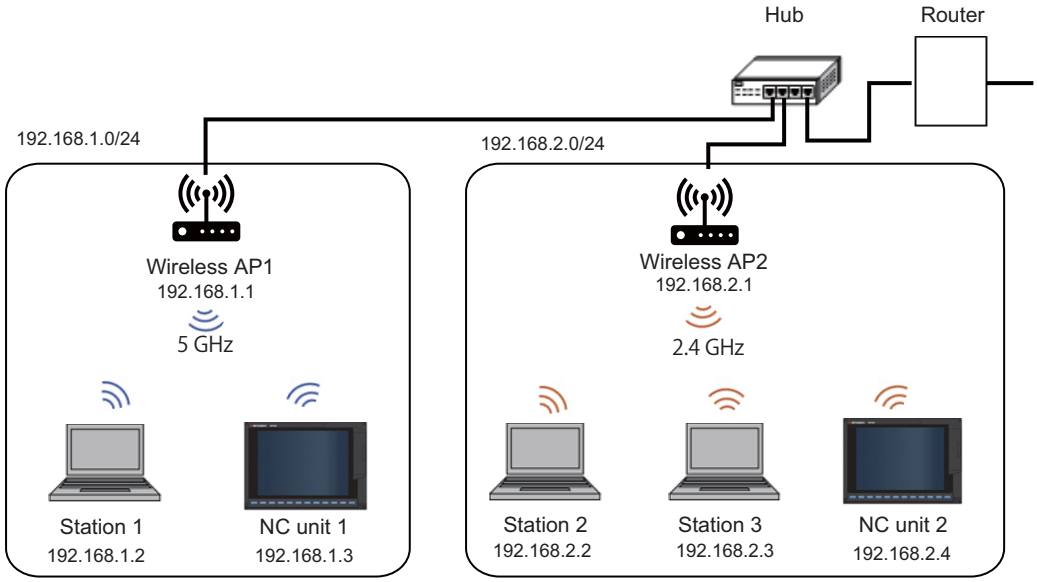
- #75001 WLAN IP address
- #75002 WLAN Subnet mask
- #75003 connect SSID
- #75007 Encryption key

(3) Ensure that the wireless access point is in such a state that access from stations is allowed.

(4) Reboot the NC and wait until the  icon indicating STA mode operation (Connected) is displayed to the left of the NC status. Setting has completed when the icon is displayed.

Example of network configuration

The following shows an example of network configuration when the NC unit operates as a station (STA).



[Configuration example of wireless LAN parameters]

Parameters		Setting of NC unit 1	Setting of NC unit 2
75000	WLAN STA/AP mode	1	1
75001	WLAN IP address	192.168.1.3	192.168.2.4
75002	WLAN Subnet mask	255.255.255.0	255.255.255.0
75003	connect SSID	ACCESSPOINT-1	ACCESSPOINT-2
75007	Encryption key	ACCESSPOINT1-KEY	ACCESSPOINT2-KEY

[Configuration example of wireless access point]

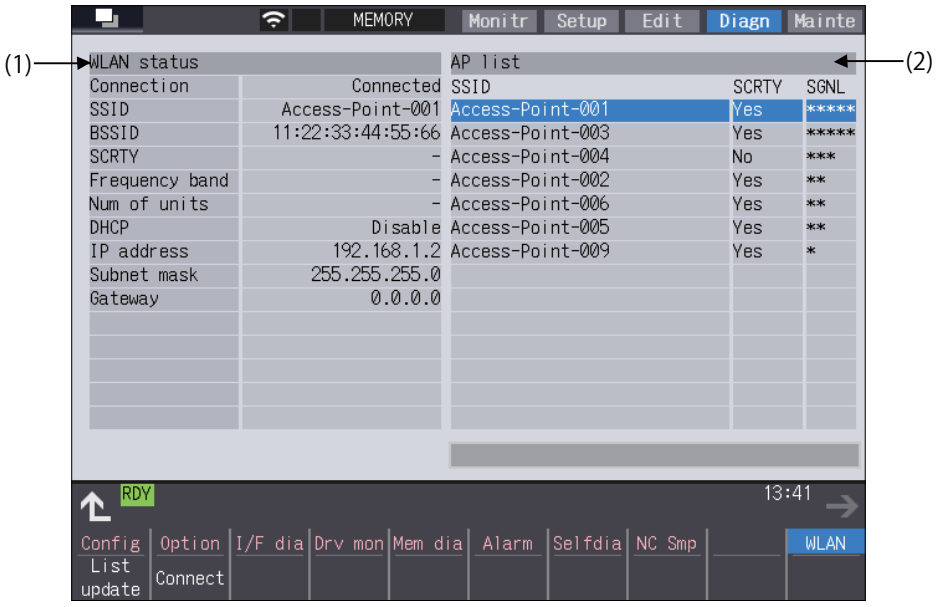
Item	Setting of Wireless AP 1	Setting of Wireless AP 2
IP address	192.168.1.1	192.168.2.1
Subnet mask	255.255.255.0	255.255.255.0
Network name (SSID)	ACCESSPOINT-1	ACCESSPOINT-2
Security mode	WPA-PSK	WPA2-PSK
Encryption type	TKIP	AES
Encryption key	ACCESSPOINT1-KEY	ACCESSPOINT2-KEY

15.2 Wireless LAN Diagnosis Screen

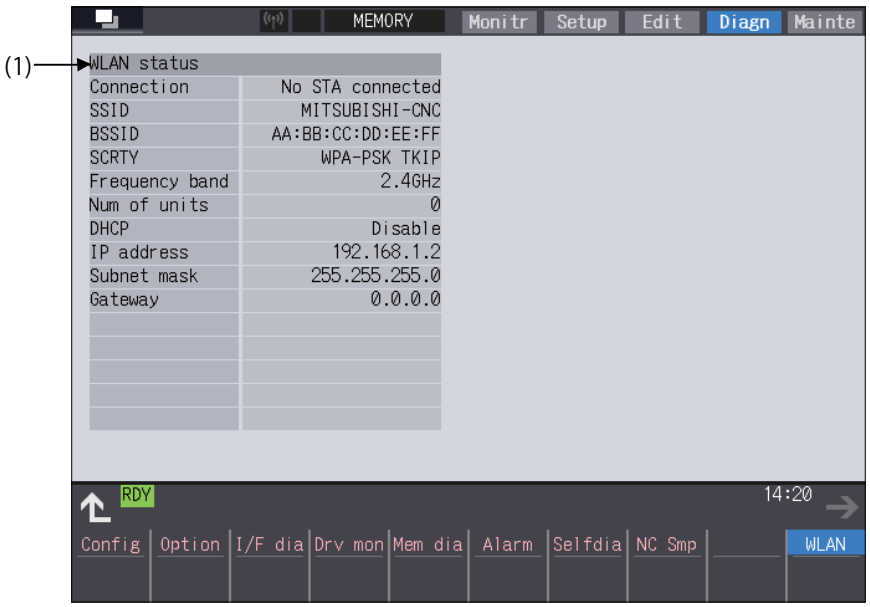
Diagn – WLAN

The wireless LAN diagnosis screen is a screen to display the operating status of wireless LAN and set the connection access point (AP).

<Station (STA) mode>



<Access point (AP) mode>



Display items

<table>
<tr><th colspan="3">Display items</th><th colspan="4">Details</th></tr>
<tr><td colspan="2">(1) WLAN status</td><td colspan="4">A list of the operating status of wireless LAN is shown. The display is as follows depending on the operating status of wireless LAN.</td></tr>
<tr><td colspan="2"></td><td colspan="2">Station (STA) mode</td><td colspan="2">Access point (AP) mode</td></tr>
<tr><td></td><td>Connection</td><td colspan="2">The connection status with the current access point is displayed.</td><td colspan="2">The connection status with the current station is displayed.</td></tr>
<tr><td></td><td></td><td>Display</td><td>Status</td><td>Display</td><td>Status</td></tr>
<tr><td></td><td></td><td>Disconnected</td><td>Not connected to the access point.</td><td>No STA connected</td><td>Not connected to the station.</td></tr>
<tr><td></td><td></td><td>Connected</td><td>Connected to the access point.</td><td>STA connected</td><td>Connected to one or more stations.</td></tr>
<tr><td></td><td></td><td>Error</td><td>A failure occurred in the wireless LAN communication module, and communication is disabled.</td><td>Error</td><td>A failure occurred in the wireless LAN communication module, and communication is disabled.</td></tr>
<tr><td></td><td>SSID</td><td colspan="2">The SSID of the connection access point is displayed. It is not displayed when it cannot be obtained from the access point.</td><td colspan="2">The SSID set in the parameter "#75005 SSID" is displayed.</td></tr>
<tr><td></td><td>BSSID</td><td colspan="2">The BSSID of the connection access point is displayed. It is not displayed when it cannot be obtained from the access point.</td><td colspan="2">The MAC address of the wireless LAN communication module built in the CNC unit.</td></tr>
<tr><td></td><td>SCRTY</td><td colspan="2">Not displayed.</td><td colspan="2">Wireless communication data encryption and authentication method is displayed.</td></tr>
<tr><td></td><td></td><td colspan="2"></td><td>Display</td><td>Meaning</td></tr>
<tr><td></td><td></td><td colspan="2"></td><td>WPA-PSK TKIP</td><td>The authentication method is WPA-PSK, and the encryption method is TKIP.</td></tr>
<tr><td></td><td></td><td colspan="2"></td><td>WPA2-PSK AES</td><td>The authentication method is WPA2-PSK, and the encryption method is AES.</td></tr>
<tr><td></td><td>Frequency band</td><td colspan="2">Not displayed.</td><td colspan="2">The frequency band (2.4 GHz/5 GHz) set in the parameter "#75009 Frequency band" is displayed.</td></tr>
<tr><td></td><td>Num of units</td><td colspan="2">Not displayed.</td><td colspan="2">The number of connected stations is displayed.
This includes the terminals where data communication is blocked by the IP address filter.</td></tr>
<tr><td></td><td>DHCP</td><td colspan="2">The client state set in the parameter "#75050 DHCP client" is displayed.
0: Disable 1: Enable</td><td colspan="2">The DHCP server state (Enable/Disable) is displayed.</td></tr>
<tr><td></td><td>IP address</td><td colspan="2"><When the DHCP client is disabled>
The value set in the parameter "#75001 WLAN IP address" is displayed.
<When the DHCP client is enabled>
The IP address obtained from the DHCP server is displayed. It is not displayed when it cannot be obtained from the DHCP server.</td><td colspan="2">The IP address set in the parameter "#75001 WLAN IP address" is displayed.</td></tr>
<tr><td></td><td>Subnet mask</td><td colspan="2"><When the DHCP client is disabled>
The value set in the parameter "#75002 WLAN Subnet mask" is displayed.
<When the DHCP client is enabled>
The subnet mask obtained from the DHCP server is displayed. It is not displayed when it cannot be obtained from the DHCP server.</td><td colspan="2">The value set in the parameter "#75002 WLAN Subnet mask" is displayed.</td></tr>
<tr><td></td><td>Gateway</td><td colspan="2"><When the DHCP client is disabled>
The value set in the parameter "#1928 Global Gateway" is displayed.
<When the DHCP client is enabled>
The gateway IP address obtained from the DHCP server is displayed. It is not displayed when it cannot be obtained from the DHCP server.</td><td colspan="2">The value set in the parameter "#1928 Global Gateway" is displayed.</td></tr>
</table>

Display items		Details
(2) AP list (*1)		The access points found with the access point search (scan) are listed in the order of stronger signal strength.
	SSID	The SSID of the access point is displayed.
	SCRTY	Whether or not security is set is displayed.
	SGNL	The signal strength is displayed in 5 levels with "*". ***** (Very strong) to * (Very weak)

(*1) These items are displayed only when the operation mode is the station (STA) mode.
Up to 14 access points are displayed in the list. Even when 15 or more access points are found by the scan, only 14 access points are displayed. For 15-type/19-type display unit, up to 16 access points are displayed.

Menus

Menus	Details
List update	This scans access points near the CNC unit, and displays available access points in the list. <Note> ♦This menu is not displayed when the operation mode is the access point (AP) mode.
Connect	This sets the access point selected in the AP list as the connection destination. <Note> ♦This menu is not displayed when the operation mode is the access point (AP) mode.

15.2.1 Setting the Access Point (AP) Nearby as Connection Destination

Operation method (Setting the access point as connection destination)

This scans access points near the CNC unit, and displays available access points in the list. Select the target access point from the list, and set it as the connection destination.

(1) Press the menu [List update]. → The network scan starts. The operation message "List updating" is displayed in flickering during the scan. After the scan is completed, the operation message "List update complete" appears, and the scan result is displayed in the list.

(2) Press the [↑] or [↓] key to move the cursor to the access point which is to be set as the connection destination.

(3) Press the menu [Connect]. → The operation message "Enter the encryption key" appears.

<Note>

- When an access point with no security settings is selected, the operation message "Cannot connect to an unsecured AP." appears, and the connection cannot be made.

(4) Enter the encryption key, and press the [INPUT] key. → The selected access point is set as the connection destination. To enable the setting, turn the power ON again.

Note

(1) If no access point is found when the scan is completed, the menu [Connect] cannot be selected.
(2) If an invalid value is entered in the encryption key, the operation message "Setting error" appears, and the connection destination is not set.
(3) The AP list found by the access point search (scan) is held until the power is turned ON again.
(4) When you enter the encryption key, "*" appears in the input area.

Operation method (Interrupting the access point search)

To interrupt the access point search, press the menu [List update] again while the scan is performed ("List updating" is displayed in flickering). Also when the  button is pressed, the scan can be interrupted.

15.3 Description of Advanced Functions

15.3.1 DHCP

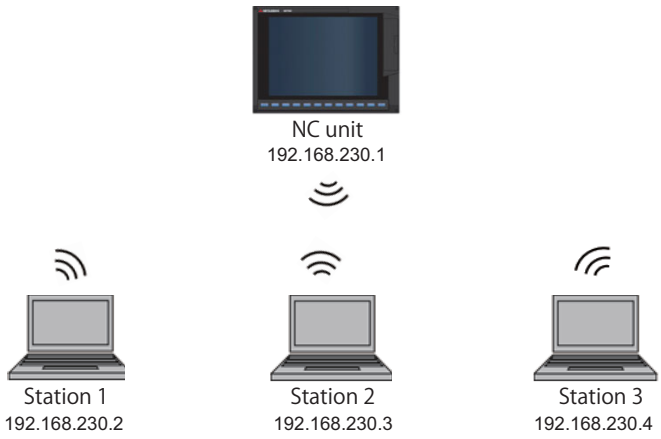
The DHCP function automatically assigns the configuration information (e.g. IP address) required for joining the network. It eliminates the need to manually assign IP addresses, reducing tasks required for network management.
The NC unit supports both the DHCP server function (automatically assigns the configuration information of the network to DHCP clients) and the DHCP client function (automatically obtains the configuration information of the network from the DHCP server).

DHCP server

The DHCP server function automatically assigns the configuration information (e.g. IP address) required for joining the network to a DHCP client when the NC unit operates as an access point (AP). To enable this function, set the parameter "#75051 DHCP server" to "1". Then, configure the following parameters to suit your network environment.

- #75052 StartIP DHCPserver
- #75053 IP num DHCPserver

The following shows an example of network configuration when the NC unit operates as DHCP server.



[Configuration example of wireless LAN parameters]

Parameters		Configuring NC unit
75000	WLAN STA/AP mode	2
75001	WLAN IP address	192.168.230.1
75002	WLAN Subnet mask	255.255.255.0
75051	DHCP server	1
75052	StartIP DHCPserver	192.168.230.2
75053	IP num DHCPserver	253

[Configuration example of wireless station]

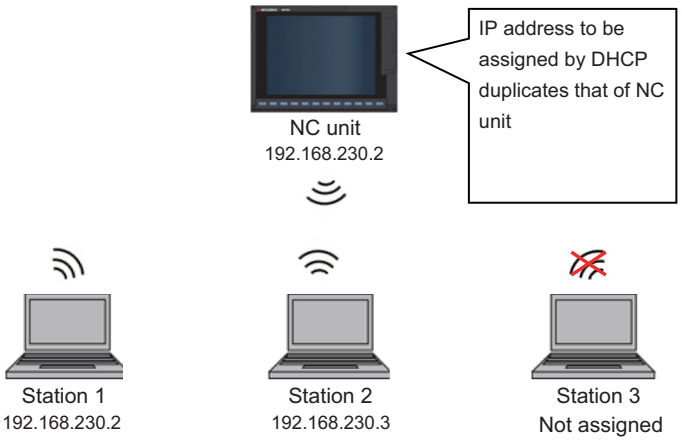
Item	Settings for Wireless AP 1
IP address	Automatic assignment
Subnet mask	Automatic assignment
DHCP client	Enabled

■Example: The number of DHCP assigned addresses becomes the value less by one than the setting value

On using DHCP server, if any of IP addresses which the client provides duplicates the value set by the parameter "#75001 WLAN IP address", the number of DHCP assigned addresses becomes the value less by one than the setting value of the parameter "#75053 IP num DHCPserver".

The following shows an example of network configuration when the number of DHCP assigned addresses becomes the value less by one the setting value.

Because IP address is not assigned to station 3, it cannot communicate with the access point



[Configuration example of wireless LAN parameters]

Parameters		Configuring NC unit
75000	WLAN STA/AP mode	2
75001	WLAN IP address	192.168.230.2
75002	WLAN Subnet mask	255.255.255.0
75051	DHCP server	1
75052	StartIP DHCPserver	192.168.230.1
75053	IP num DHCPserver	3

[Configuration example of wireless station]

Item	Settings for Wireless AP 1
IP address	Automatic assignment
Subnet mask	Automatic assignment
DHCP client	Enabled

■Example: DHCP server does not work

Do not set the value of "255" or larger to the 4th octet of IP address which the client provides to both the parameter "#75052 StartIP DHCPserver" and "#75053 IP num DHCPserver". That disables the wireless LAN parameter settings.

The following shows an example of parameter settings which the DHCP server does not work.

When the 4th octet of the parameter "#75052 StartIP DHCPserver" is "253" and setting value of the "#75053 IP num DHCPserver" is "3", the 4th octet of IP address to be assigned to the 3rd station becomes "255". Thus, the wireless LAN parameter settings are disabled.

[Configuration example of wireless LAN parameters]

Parameters		Configuring NC unit
75000	WLAN STA/AP mode	2
75001	WLAN IP address	192.168.230.1
75002	WLAN Subnet mask	255.255.255.0
75051	DHCP server	1
75052	StartIP DHCPserver	192.168.230.253
75053	IP num DHCPserver	3

■Setting of the parameter "#75002 WLAN Subnet mask"

The assignable range of the IP addresses ("#75001 WLAN IP address", "#75052 StartIP DHCPserver") can be set with the parameter "#75002 WLAN Subnet mask". The following shows a setting example of the parameter. To assign a wider IP address range, set "255.255.0.0" or "255.0.0.0" in #75002.

[Setting example of the parameter #75002]

Parameters		Configuring NC unit
75001	WLAN IP address	192.168.230.1
75002	WLAN Subnet mask	255.255.255.0
75003	StartIP DHCPserver	192.168.230.2 to 192,168,230,254

To limit the range of the parameter #75052, take care of the following points to set the parameter.
Set a decimal value representable in the host part (the range where "0" is set in the bit) of the parameter #75002, in the IP address (#75001, #75052). A decimal value where the entire host part of the subnet mask is represented by "0" or "1" cannot be assigned to the IP address.

The following shows a setting example of the parameter. In the table below, #75002 is set with "0" in the low-order 5 bits; therefore, the assignable IP address range is "1 (= 0000 0001) to 30 (= 0001 1110)". In the setting example of "Unconnectable", the IP address is set out of the range of "1 to 30"; therefore, the IP address cannot be assigned to the station, and connection cannot be made.

[Setting example of the parameter where the IP address range is limited]

Parameters		Configuring NC unit	
		Unconnectable	Connectable
75001	WLAN IP address	192.168.10.193 (193 = 1100 0001)	192.168.10.1 (1 = 0000 0001)
75002	WLAN Subnet mask	255.255.255.224 (224 = 1110 0000)	255.255.255.224 (224 = 1110 0000)
75052	StartIP DHCPserver	192.168.10.200 (200 = 1100 1000)	192.168.10.2 (2 = 0000 0010)
75053	IP num DHCPserver	20	20

DHCP client

The DHCP client function automatically obtains the configuration information (e.g. IP address) required for joining the network from the DHCP server when NC unit operates as a station. To enable this function, set the parameter "#75050 DHCP client" to "1".

[Configuration example of wireless LAN parameters]

Parameters		Configuring NC unit
75000	WLAN STA/AP mode	1
75050	DHCP client	1

[Configuration example of wireless access point]

Item	Settings for Wireless AP 1
IP address	192.168.1.1
Subnet mask	255.255.255.0
DHCP server	Enabled
IP address to allocate	192.168.1.2
Number of allocations	64

15.3.2 ESSID Stealth

The ESSID stealth function prevents unidentified stations from accessing the NC unit operating as an access point (AP).

Usually, a wireless access point (AP) transmits a beacon to announce its ESSID to stations. With the ESSID stealth function enabled, no beacon is transmitted so that the ESSID is not broadcast to stations. To enable this function, set parameter "#75010 ESSID stealth" to "1".

With the ESSID stealth function disabled, the ESSID of the NC unit is broadcast to the wireless devices within the area covered by beacon transmitted from the NC unit.

15.3.3 AP Isolation

The AP isolation function prevents stations from communicating with each other when the NC unit operates as an access point (AP). This function may also be called privacy separator function. This function is enabled by default. To disable this function, set parameter "#75011 AP isolation OFF" to "1".

With the AP isolation function enabled, communication between stations cannot be made through the NC unit.

15.3.4 Abg Mode

The abg mode function restricts communication as per IEEE802.11n when the NC unit operates as an access point (AP). This function is used when communication with a device operating as a wireless station does not work properly for reasons such as compatibility when IEEE802.11n is applied. To enable this function, set parameter "#75012 abg mode" to "1".

15.3.5 BSSID

The BSSID function enables to connect with a specific AP by specifying the AP's MAC address. BSSID is used when a specific AP needs to be used in roaming, for example.

To use this function, set the SSID of an access point (AP) in the parameter "#75003 connect SSID", and the MAC address of the the access point (AP) in the parameter "#75004 connect BSSID". If SSID and BSSID do not match the AP, connection cannot be made.

15.4 Relationship with Other Functions

IP filter setting

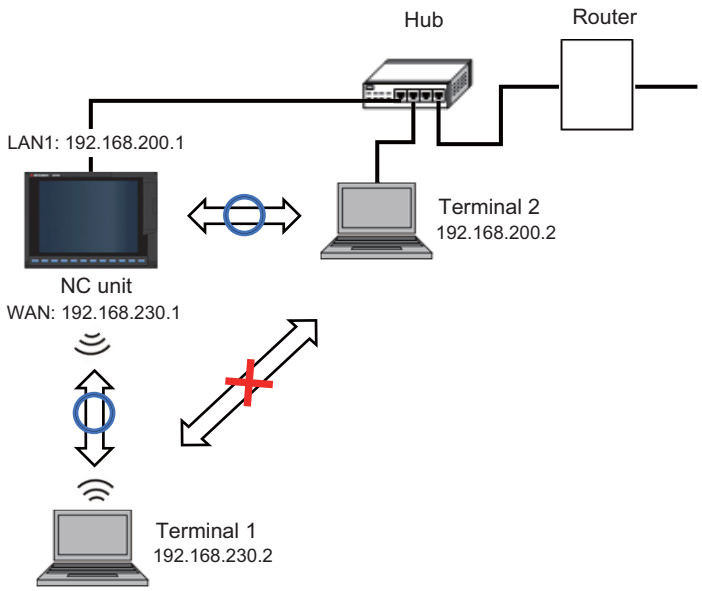
The IP filter setting function blocks unauthorized access from external devices by identifying the IP addresses of the accessing device. For more details on the IP filter setting function, refer to "Appendix 3: IP Filter Setting Function".

Wired LAN

Although the NC unit supports wired connections using Ethernet cables, it cannot relay communications between wireless communication and wired communication because it handles them separately.

The following is an example of how to communicate between a wireless and wired connection.

Because the NC unit does not relay communications between the wireless LAN network (192.168.230.0/24) and the wired LAN network (192.168.200.0/24), Terminal 1 within the wireless LAN network and Terminal 2 within the wired LAN network cannot communicate with each other.



Gateway

Although the wired LAN port and the wireless LAN port of the NC unit belong to different subnets, only one IP address can be set for the gateway. As such, if an IP address in the subnet of the wired LAN is set in parameter "#1928 Global Gateway", the NC unit cannot communicate with devices in a different subnet over the wireless LAN network. When the NC unit is to communicate with a device in a different subnet using wireless LAN, specify an IP address in the subnet of wireless LAN in parameter "#1928 Global Gateway".

15.5 Troubleshooting

Symptom	Corrective action
No wireless communication	[When the icons indicating the operating status of wireless LAN () are not displayed] •Verify that the wireless LAN parameters have been configured correctly. •Verify that the country/region code is set. If not, the wireless LAN functionality is disabled even if the wireless LAN parameters have been configured correctly. [When an icon indicating the operating status of wireless LAN is displayed] •Verify that there are no conflicting IP addresses. •Verify that communication is not blocked by functions such as IP filter setting or MAC address filtering of the wireless router. •Changing the distance or angle of a communication device may improve connection status. •Verify that the wireless LAN channel that corresponds to the country/region code has been configured correctly.
Wireless communication is interrupted or communication speed is slow/unstable	•Move the NC unit closer to the communicating device and see if communication is stabilized. If there is an obstruction nearby, remove or move it so that radio waves reach easily. •If a device operating near the NC unit transmits radio wave in the same frequency band as the NC unit, wireless communication may become unstable. Increase the distance between the device and the NC unit, or stop using the device temporarily. If it is impossible, change the setting of the parameter "#75009 Frequency band" of the NC unit to avoid radio frequency interference. •Set the wireless LAN channel avoiding the frequency interference nearby. Automatic channel setting enables select a channel suitable for the frequency status of your neighborhood. •If communicating device is located on the back side of the machine, the communication might be unstable. Change the location of the communicating device to other position.
The DHCP server does not work	•Verify that the settings for DHCP server in wireless LAN parameters have been configured correctly. •The DHCP server function and the IP filter setting function for wireless LAN cannot be used together. When using the DHCP server function, set the parameter "#75100 IP Filter for WLAN" to "0" to disable the IP filter setting function for wireless LAN. •If any of IP addresses which the client provides duplicates the value set by the parameter "#75001 WLAN IP address", the number of DHCP assigned addresses becomes the value less by one than the setting value of the parameter "#75053 IP num DHCPserver". Set the larger value to #75053 or change the value of "#75052 StartIP DHCPserver" to that not included in the address provided by #75001. For details, refer to "15.3.1 DHCP". •Ensure that "#75052 StartIP DHCPserver" and "#75053 IP num DHCPserver" are not configured in such a way that a value greater than or equal to 255 is assigned to the 4th octet of the IP addresses provided by the DHCP server. Change the settings of #75052 and #75053 not to include "255" in IP addresses. For details, refer to "15.3.1 DHCP".

15.6 The Software License Authorized for This Function

wpa_supplicant and hostapd

Copyright (c) 2002-2019, Jouni Malinen <j@w1.fi> and contributors All Rights Reserved.

These programs are licensed under the BSD license (the one with advertisement clause removed).

If you are submitting changes to the project, please see CONTRIBUTIONS file for more instructions.

This package may include either wpa_supplicant, hostapd, or both. See README file respective subdirectories (wpa_supplicant/README or hostapd/README) for more details.

Source code files were moved around in v0.6.x releases and compared to earlier releases, the programs are now built by first going to a subdirectory (wpa_supplicant or hostapd) and creating build configuration (.config) and running 'make' there (for Linux/BSD/cygwin builds).

License

This software may be distributed, used, and modified under the terms of BSD license:

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

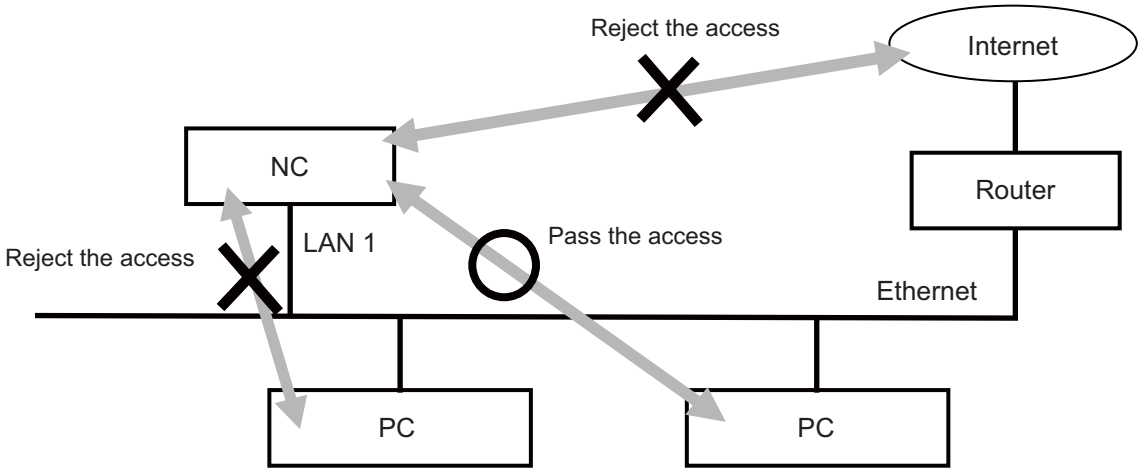
(1) Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.
(2) Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.
(3) Neither the name(s) of the above-listed copyright holder(s) nor the names of its contributors may be used to endorse or promote products derived from this software without specific prior written permission.

THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT OWNER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

16

Appendix 3: IP Filter Setting Function

This function prevents unauthorized access from external devices by filtering the IP address of the access source when the NC is being connected to Ethernet. By setting the IP addresses to be passed or rejected, access from external devices can be restricted.



16.1 Setting the IP Address Filter

To enable the IP filter setting function, set the relevant parameters and then turn the power ON again.
You can set IP addresses for each LAN communication port of an NC whether to permit or restrict the access.

The restriction for the access is set by the parameter #8910 or #9830.

Setting value	Restriction	Details
0	Disabled	Disable the IP filter setting function.
1	Pass	When the range of IP addresses from which access to the NC should be allowed is set, access from an IP address within the range is permitted. Access from the other IP addresses is restricted.
2	Reject	When the range of IP addresses to block is set, access from an IP address within the range is blocked. Access from the other IP addresses is permitted.

Set the range of IP addresses (starting and ending IP addresses) to the following parameters.
Up to 8 ranges of IP addresses can be set.

LAN port and setting range of the parameters for the respective model

Connector name		Parameter No.	Details
M800VW/M80VW Series	M800VS/M80V Series		
LAN	LAN1	#9810	IP filter setting for LAN1 0: Invalid 1: Pass 2: Reject
		#9811/#9812 to #9825/#9826	Starting/ending IP filter setting LAN1-1 to LAN1-8
GDI	-	#9830	IP filter setting for LAN2 0: Invalid 1: Pass 2: Reject
		#9831/#9832 to #9845/#9846	Starting/ending IP filter setting LAN2-1 to LAN2-8
LAN3	-	#9850	IP filter setting for LAN3 0: Invalid 1: Pass 2: Reject
		#9851/#9852 to #9865/#9866	Starting/ending IP filter setting LAN3-1 to LAN3-8
-	WLAN	#75100	IP filter setting for WLAN 0: Invalid 1: Pass 2: Reject
		#75101/#75102 to #75115/#75116	Starting/ending IP filter setting WLAN-1 to WLAN-8

16.2 Examples for Using IP Address Filter

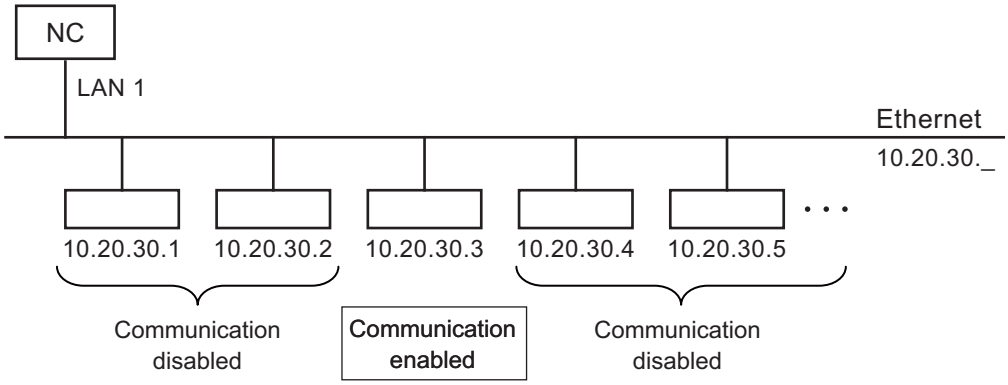
■To permit (pass) the access from only one device

Set the same value to the starting and ending IP addresses.

<Parameter settings>
#9810 IP Filter for LAN1: 1
#9811 StartFiltIP LAN1-1: 10.20.30.3
#9812 EndFilterIP LAN1-1: 10.20.30.3

<Operation>

Access to the NC from "10.20.30.3" is permitted, and access from the other IP addresses is restricted.

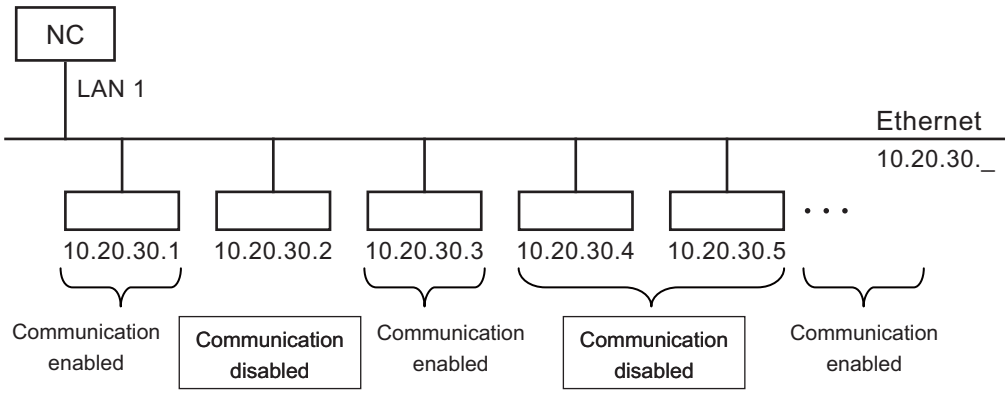


■To restrict (reject) the access from multiple ranges of IP addresses, and to permit the access from the other IP addresses

<Parameter settings>
#9810 IP Filter for LAN1: 2
#9811 StartFiltIP LAN1-1: 10.20.30.2
#9812 EndFilterIP LAN1-1: 10.20.30.2
#9813 StartFiltIP LAN1-2: 10.20.30.4
#9814 EndFilterIP LAN1-2: 10.20.30.5

<Operation>

Access to the NC from "10.20.30.2", and from the range of "10.20.30.4" to "10.20.30.5" is restricted, and access from the other IP addresses is permitted.

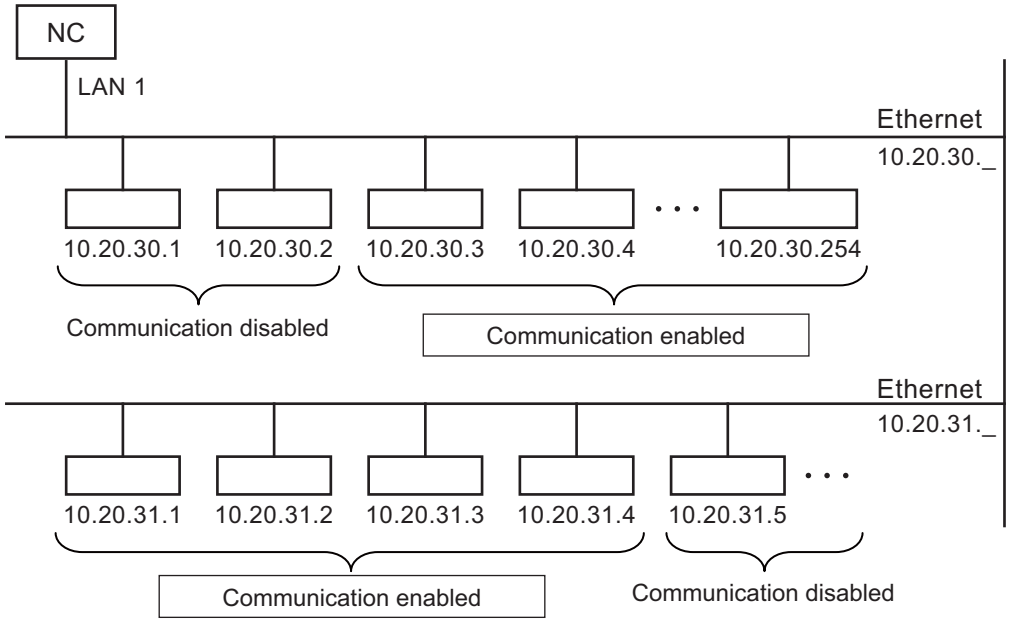


■To permit (pass) the access from multiple ranges of IP addresses, and to restrict the access from the other IP addresses

<Parameter settings>
#9810 IP Filter for LAN1: 1
#9811 StartFiltIP LAN1-1: 10.20.30.3
#9812 EndFilterIP LAN1-1: 10.20.31.4

<Operation>

Access to the NC from the range of "10.20.30.3" to "10.20.30.254" and "10.20.31.1" to "10.20.31.4" is permitted, and access from the other IP addresses is restricted.



Precautions

(1) Do not restrict the access from the IP address of the separate type display unit. This causes that the NC cannot communicate with the display unit. If the setting is already made, perform the steps (a) to (e) below to solve the problem.
  (a) While the NC power is being OFF, set the rotary switch of the control unit to "B", and then turn the NC power ON.
  (b) After the display of 7-segment LEDs changes "0y", turn the NC power OFF.
  (c) Turn the rotary switch to "0" again.
  (d) Turn the NC power ON.
  (e) By performing the steps (a) to (d), the NC restarts with the IP filter setting function temporarily disabled. Reset the parameter setting for IP filter to an appropriate value.

  If you turn the NC power OFF without changing the parameter setting in the step (e), the remedy should be performed from step (a) again.

(2) For the setting of IP addresses, it need not be in ascending order. Even though the value of the starting IP address is larger than that of the ending IP address, the addresses in the setting range are targets for the IP filter setting function.

(3) When there is the range which includes either the starting or ending IP address set to "0.0.0.0", "Y05 Initial parameter error" is displayed.

(4) When the IP filter setting function is enabled ("Pass" or "Reject" is selected) with no valid IP address range, "Y05 Initial parameter error" is displayed and IP filter setting function is disabled.

(5) "0.0.0.0" cannot be used for the valid IP address because this address has special meaning.
Broadcasting IP address (end of the address is "255") cannot be used for the valid IP address either.

(6) Even if the range which includes the IP address of an NC is set to prohibit the access, communication with the NC can be made properly.

17

Appendix 4: VNC Server Function

VNC server function enables status confirmation and remote operation by displaying the NC screen display on the external computer/tablet type computer.
In order to use this function, the unit needs to have an optional specification "VNC server function" and the parameter "#19701 Restrain VNCserver" needs to be set to "1" or "2".

17.1 Environment Establishment (Preparation)

The parameter needs to be set in order to connect to VNC client.

Setting Ethernet parameters

Use the parameters #1953 to #1955. Set the same group IP address to "#1953 Intra IP address" as the external computer/tablet type computer in which VNC client is operated.

Note

(1) For M80V Series, set the same values to the parameters "#1926 Global IP address" to "#1928 Global Gateway".
(2) For M800VS Series, you need to set the parameters "#1926 Global IP address" and "#1953 Intra IP address" to the IP address used in the same group. Set the same value to #1954 and #1955 as #1927 and #1928. When the different value is set, the connection cannot be made.

Setting operation parameters

Set a password to "#19702 VNC password".

"0" (1 character) cannot be set as a password. When "0" is input, the set password is cleared.

Set a port No. to "#19703 VNC server port".

Set "5901" as recommended value.

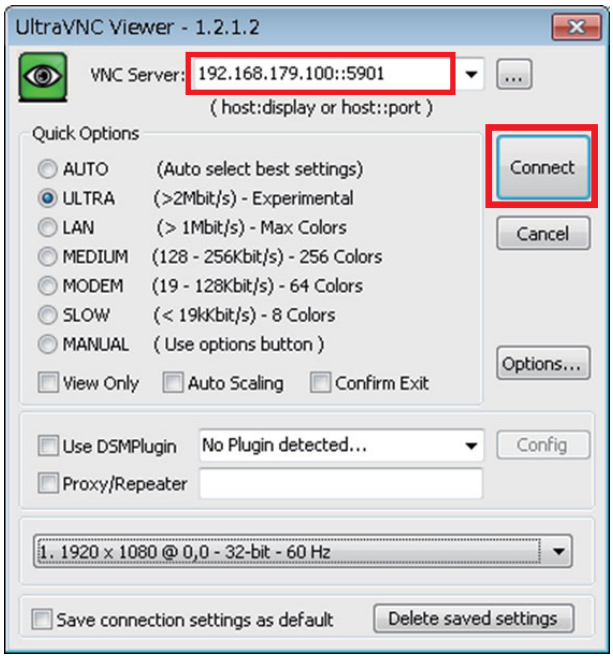
17.2 Displaying the NC Screen with VNC Client

Use "UltraVNC Viewer" as NC client.

17.2.1 Connecting VNC Client

(1) Click "UltraVNC" - "UltraVNC Viewer" from the start menu to start VNC client.

(2) Input the IP address of VNC server set in the parameter "#1953 Intra IP address" and the port No. set in "#19703 VNC server port" by connecting with "::", and click [Connect].



(3) When the password input screen is displayed, input the password set in the parameter "#19702 VNC password" and then click [Log On].

(4) VNC server is connected and NC screen is displayed.

The settings of "UltraVNC Viewer" are as follows.

Display items		Details
Color depth setting	AUTO	Set ULTRA to SLOW.
	ULTRA	Connect according to the setting of the parameter "#19704 VNC color depth". (recommended setting)
	LAN	
	MEDIUM	Connect in 8bit color. Display in the different color shade from NC screen.
	MODEM	Connect in 6bit color. Display in the different color shade from NC screen.
	SLOW	Connect in 3bit color. Display in the particularly different color shade from NC screen.
	MANUAL	Click [Options…] to set.
View Only		The operation from VNC client cannot be performed. Do not check this box.
Auto Scaling		Expand the VNC client screen according to the screen of external computer. Set this as necessary.
Confirm Exit		Display the confirmation message when VNC client is ended. Set this as necessary.
Use DSMPlugin		Setting for communication encryption. Do not check this box as VNC server function does not support encryption.
Proxy/Repeater		Setting to connect VNC client to the NC of a factory network from the external network. Do not check this box as VNC server function does not support the connection using Proxy.
Save connection settings as default		Save the settings as default. Set this as necessary.

Note

(1) When operating VNC client and actual NC unit simultaneously, the operation may not performed as expected.

(2) Even when the NC is turned OFF while connecting to VNC client, VNC client keeps communicating with NC while displaying the screen at the time of power OFF in order to update the display. In that case, close VNC client once and then turn ON the NC again before connecting to VNC client again.

(3) While connecting to VNC client, if you attempt to connect to "UltraVNC Viewer", "Connecting…" keeps displaying and the connection cannot be made.

(4) When VNC client is operated on the computer using wireless network, if the state that the operation is not performed for a certain period of time with VNC client, the screen update may be stopped. By operating VNC client, the screen update may be recovered.

17.2.2 Operating the NC Using the Keyboard

The key operation on the NC screen can be performed from the PC keyboard of VNC client.

The following table shows the correspondence between NC keyboard and PC keyboard.

NC keyboard	PC keyboard	Supplement	NC keyboard	PC keyboard	Supplement
(MONITOR)	Shift + F1	Monitor screen (Monitr)	MENU8	F8	
(SETUP)	Shift + F2	Setup screen (Setup)	MENU9	F9	
(EDIT)	Shift + F3	Edit screen	MENU10	F10	
(DIAGN)	Shift + F4	Diagnose screen (Diagn)	◁	F11	Return (menu)
(MAINTE)	Shift + F5	Maintenance screen (Mainte)	▷	F12	Next menu
$↔$	Ctrl + F1	System switching	DELETE	Delete	DELETE
?	Ctrl + F2	Display/non-display of the operation guidance, parameter guidance and alarm guidance	INSERT	Insert	INSERT
			C.B.	Shift + Home	Clear the input area
BACK	Ctrl + F6	Select the previous screen Valid on the diagnosis or maintenance screen	CAN	Esc	
			ENTER	Enter	
LIST	Ctrl + F8	Display/non-display of menu List	↑	↑	
	Ctrl + F9	Switch to the previous tab	↓	↓	
	Ctrl + F10	Switch to the next tab	←	←	
PAGE	PageUp	Display the previous page	→	→	
PAGE	PageDown	Display the next page	Left tab (|←)	Shift + Tab	Move the cursor of input target data
MENU1	F1		Right tab (→|)	Tab	Move the cursor of input target data
MENU2	F2		A to Z	A to Z	Alphabet
MENU3	F3		0 to 9	0 to 9	Numbers
MENU4	F4		+ - / * @ , . ! ; : ( ) [ ] # $ =	+ - / * @ , . ! ; : ( ) [ ] # $ =	Function symbol
MENU5	F5				
MENU6	F6				
MENU7	F7				

Uppercase/lowercase letters at the time of alphabet input can be switched with CapsLock key or Shift key on the external computer in which VNC client is operated.

Note

(1) Even if the parameter "#19701 Restrain VNCserver" is set to "2", the operations other than Enter key from the computer and [INPUT] key on the software keyboard cannot be restrained.

17.2.3 Imposing Restrictions to VNC Client Operations

When the parameter "#19701 Restrain VNCserver" is set to "2", the operations of Enter key from the computer and [INPUT] key on the software keyboard are invalid.

17.3 Troubleshooting

The following shows the countermeasures for general troubles related to VNC client connection.

Symptom	Possible cause	Countermeasure
Cannot connect to VNC client.	LAN cable is not connected.	Connect LAN cable to LAN1 terminal of the NC unit in which VNC server is operated.
	The parameters "#1953 Intra IP address", "#1954 Intra Subnet mask" and "#1955 Intra Gateway" are not set properly.	Set the same group IP address to parameters #1953, #1954 and #1955 as the external computer/tablet type computer in which VNC client is operated.
	Password is not set in the parameter "#19702 VNC password".	Set the password to the parameter "#19702".
	The port No. set in the parameter "#19703 VNC server port" is already used elsewhere in the network.	Set the recommended value "5901" in the parameter "#19703 VNC server port".
	Another VNC client is already connected.	While already connecting to another VNC client, if you attempt to connect a new VNC client, "Connecting…" keeps displaying and the connection cannot be made. When there is already VNC client connected, disconnect it first before connecting another VNC client.
	VNC client which does not support RFB protocol 3.3, 3.7 and 3.8 is used.	Use VNC client which supports RFB protocol 3.3, 3.7 and 3.8.
	VNC client which does not support the security type "VNC Authentication" is used.	Use VNC client which supports the security type "VNC Authentication".
	VNC client application terminated illegally.	If a new connection cannot be made after VNC client application is terminated in the external computer, it may be re-connected by re-starting the computer.
	[For M80V Series] The parameters "#1926 Global IP address", "#1927 Subnet mask" and "#1928 Global Gateway" are not set to the same values as "#1953 Intra IP address", "#1954 Intra Subnet mask" and "#1955 Intra Gateway".	Connection cannot be made unless #1926 to #1928 are set to the same values as #1953 to #1955.
	[For M800VS Series] The parameters "#1926 Global IP address" and "#1953 Intra IP address" are not set to the IP address used in the same group.	#1926 and #1953 need to be set to the IP address used in the same group. Set #1954 and #1955 to the same values as #1927 and #1928. When the different value is set, the connection cannot be made.
The screen displayed with VNC client is different from the actual colors.	The parameter "#19704 VNC color depth" is set to "1".	When #19704 is set to "1", the displayed color is reduced to 8bit color. In order to display as same as the NC screen color, set #19704 to "0" and select "ULTRA" in the "UltraVNC Viewer" connection setting.
	"MEDIUM", "MODEM" or "SLOW" is selected in the "UltraVNC Viewer" connection setting.	When "MEDIUM", "MODEM" or "SLOW" is selected, the displayed color is reduced to 8bit or less color. In order to display as same as the NC screen color, select "ULTRA" and set the parameter "#19704 VNC color depth" to "0".

Symptom	Possible cause	Countermeasure
The operation from VNC client cannot be performed.	[View Only] is checked in the "UltraVNC Viewer" connection setting.	As the operation from VNC client cannot be performed at all with [View Only] checked, remove the check.
	The parameter "#19701 Restrain VNCserver" is set to "2".	When the parameter #19701 is set to "2", the operations of Enter key on the computer and [INPUT] key on the software keyboard are invalid. In order to enable the operations, set #19701 to "1".
	-	Gesture operations using two fingers (enlarging or reducing of 3D check shape, drag operations such as rotation, etc.) or the operations by holding down (long tapping) (displaying the parameter guidance, etc.) cannot be performed regardless of the parameter settings.
The update of VNC client display is slow.	-	By changing the setting of the parameter "#19705 VNC transmit cycle" to "1" or "2", the update may get faster. However, if the setting of #19705 is changed to other than "0", the screen update on NC unit may be slow.
	-	By changing the parameter "#19704 VNC color depth" to "1" and select "ULTRA" in the "UltraVNC Viewer" connection setting, the update may get faster. However, if the setting of #19704 is changed to "1" and "ULTRA" is selected in the "UltraVNC Viewer" connection setting, the displayed color is reduced on VNC client, which is different from NC screen color.

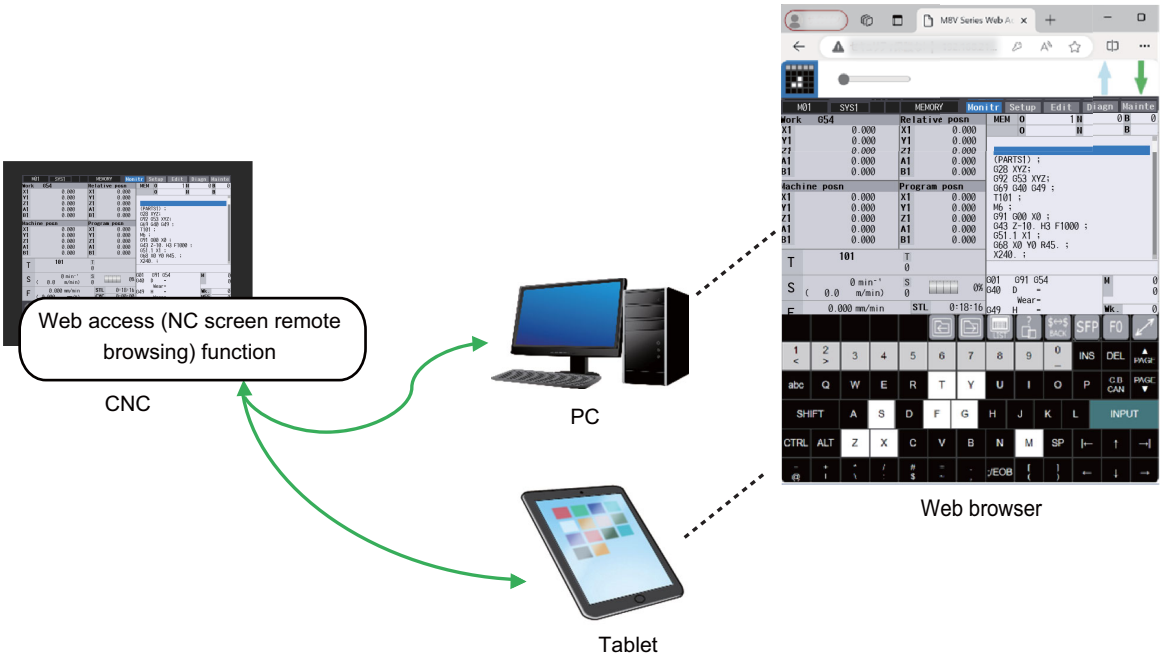
18

Appendix 5: Web Access (NC Screen Remote Browsing) [M800VS/M80V Series]

This function enables to display the CNC screen connected to LAN in the plant, on a Web browser. With this function, CNC window operation and operation status check can be performed on a terminal (such as personal computer, tablet, and smartphone) away from the machine.
This function is available on M800VS/M80V Series, and it can be used both wired and wireless.

Note

(1) Wireless LAN connection is not available for M800VS when CNC is in the station (STA) mode and the DHCP client is enabled.



■Supported OS/browser

To use this function, a Web browser supporting JavaScript is required.

Operation on the following OS and Web browser has been checked. (As of January, 2025)

OS	Web browser/version
Windows 11	Edge 128.0.2739.42
Android 11	Google Chrome 127.0.6533.106
iPadOS 17.6.1	Safari 17.6
macOS Sonoma 14.6.1	Safari 17.6

18.1 Setting for Web Access

18.1.1 Setting Parameters

To use this function, set the following parameters.

- Set "#1766 cfgPR16/bit7" (WebAccess) to "1".
- Set "#19730 WebAccess" to any of "1" to "3".

Set the user name and password in the following parameters.

- #19731 WebAccess USER
- #19732 WebAccess PSWD

18.1.2 Setting the IP Address

The setting contents differ depending on the method of connecting a terminal to display the Web browser.

■To connect wired to the LAN connector of the display unit

[M800VS Series]

Use the parameter settings of "#1953 Intra IP Address" to "#1955 Intra Gateway".
Set the IP address of CNC in the parameter "#1953 Intra IP Address". Set the IP address in the same group as the terminal.

[M80V Series]

Use the parameter settings of "#1926 Global IP Address" to "#1928 Global Gateway".
Set the IP address of CNC in the parameter "#1926 Global IP Address". Set the IP address in the same group as the terminal.

■To connect wireless to CNC

The wireless LAN parameters need to be set. For details, refer to "15.1 Configuring the Wireless LAN".

18.2 Displaying the CNC Screen on Web Browser

The following is the procedure to display the CNC screen on Web browser.

(1) Input the following URL in the address bar of the Web browser.
http:// (IP address of the connected CNC) /remote/
In "IP address of the connected CNC", input the IP address by referring to below.

- To connect wired to the LAN connector of the display unit
  M800VS Series: IP address set in the parameter "#1953 Intra IP address"
  M80V Series: IP address set in the parameter "#1926 Global IP address"
- To connect wireless to CNC
  M800VS Series: IP address where "1" is added to the address in "IP address" on the wireless LAN diagnosis screen (*1)
  M80V Series: IP address in "IP address" on the wireless LAN diagnosis screen
  For details, refer to "15.2 Wireless LAN Diagnosis Screen".

(*1) When the 4th octet of the IP address in "IP address" on the wireless LAN diagnosis screen is 254, that of the connected CNC is 1.
(Example) The IP address in "IP address" on the wireless LAN diagnosis screen is "192.168.1.254", specify "192.168.1.1" for the IP address of the connected CNC.

(2) Input the user name and password set with the parameters on the user authentication screen.
User name: User name set in #19731
Password: Password set in #19732

(3) When user authentication is successful, the CNC screen is displayed on the Web browser.

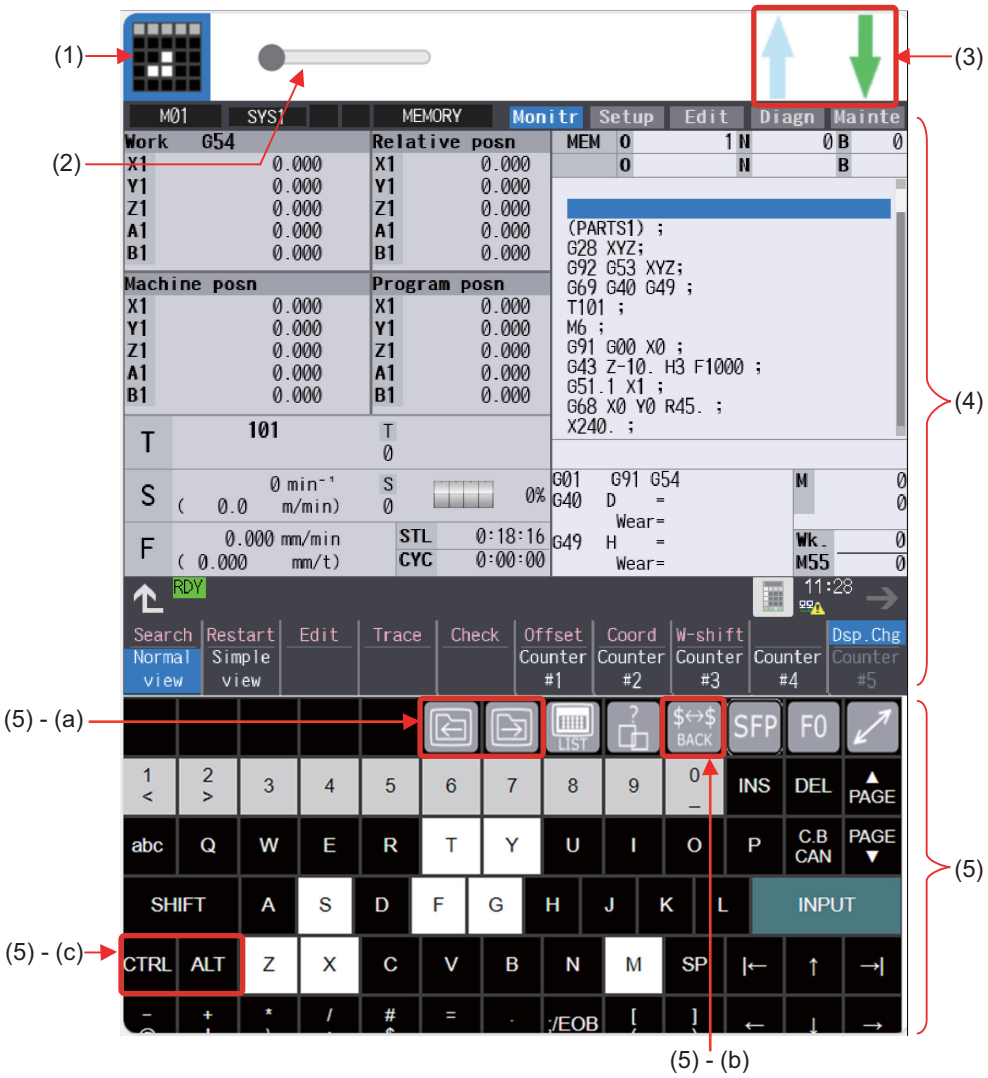
Note

(1) One CNC can be displayed on one page of the Web browser. To display another CNC, input the IP address of the desired CNC in step (1).

18.2.1 Explanations of the Screens

The displayed CNC screen is automatically adjusted to fit the width and height of the Web browser.

■Screen display example



Display items

	Display items	Details
(1)	Web keyboard show/ hide button	This changes whether to show or hide the Web keyboard. It is highlighted while the Web keyboard is displayed. The Web keyboard is automatically adjusted to fit the width of the CNC screen display area when it is displayed for the first time. For the second and subsequent times, the Web keyboard is displayed with the same width as last time. The Web keyboard may be displayed outside the Web browser display area due to size change of the Web browser display area. At this time, the Web keyboard is displayed at the initial position.
(2)	Web keyboard transparency slider	This changes the transparency of the Web keyboard. Move the slider to the right to increase the transparency of the Web keyboard. When the transparency is higher, it is easier to check the display under the Web keyboard.
(3)	Transmission/reception display	This shows the communication status. The "↑" icon is ON during transmission, and the "↓" icon is ON during reception.
(4)	CNC screen display area	The CNC screen is displayed.

Display items	Details	
(5) Web keyboard	This is used for key input on the screen. The Web keyboard can be shown/hidden, the display position can be moved, and the size can be changed. The key operation of the Web keyboard is the same as that of software keyboard. For details, refer to "2.6.2 Software Keyboard". The [INS]/[DEL] keys correspond to the [INSERT]/[DELETE] keys of the software keyboard. The following keys are not included in the software keyboard.	
	(a) Frame keys	This switches the tab.
	(b) Part system change-over key	When using a multi-part system, 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.
	(c) Operation keys	[ALT]: Alternate key [CTRL]: Control key

18.2.2 Operation Method

Reducing/enlarging the CNC screen display area

Pinch in/pinch out in the CNC screen display area to reduce/enlarge the CNC screen.

The maximum display is 400% of the initial display, and the minimum display is 10% of the initial display. The initial display magnification of the CNC screen is 100%.

Changing the display position of Web keyboard

Drag the Web keyboard to change the display position to any position on the Web browser.

Reducing/enlarging the Web keyboard

Pinch in/pinch out on the Web keyboard to reduce/enlarge the Web keyboard.

The maximum display is 400% of the initial display, and the minimum display is 10% of the initial display. The initial display magnification of the Web keyboard is 100%.

19

Appendix 6: List of Function Codes

Function code	Control unit recognition	PARITY V	Screen display	Setting and display unit	In memory	Punch-output	Internal NC system function
ISO		V count		Key-in	Store	ISO	
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 ;/EOB	Stored	(	Control out (comment start)
)	Yes	Counted	Displayed	Key-in ;/EOB	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 than the above	No	Counted	(*1)	No Key-in	Stored		

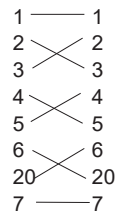
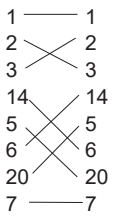
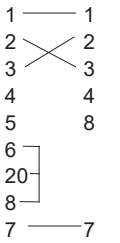
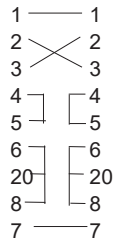
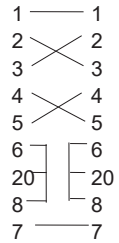
(*1) This denotes characters (including blanks) which are stored inside and which correspond to the command codes. "@" is not displayed.

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.

20

Appendix 7: RS-232C I/O Device Parameter Setting Examples

Parameter setting examples	Cable connection Parameter setting examples					
	NC I/O 	NC I/O 	NC I/O 	NC I/O 	NC I/O 	Follows communication protocol. (NC side) 2 :SD 3 :RD 4 :RS 5 :CS 6 :DR 20:ER 7 :GND
Device name						Follows communication software.
BAUD RATE	2	2	2	2	2	
STOP BIT	3	3	3	3	3	
PARITY EFFECTIVE	0	0	0	0	0	
EVEN PARITY	0	0	0	0	0	
CHR. LENGTH	3	3	3	3	3	
HAND SHAKE	3	2	3	3	3	
DC CODE PARITY	1	0	1	1	1	
DC2/DC4 OUTPUT	0	0	1	0	1	
CR OUTPUT	0	0/1	0	0	0	
FEED CHR.	0	0	0	0	0	
PARITY V	0	0	0	0/1	0/1	
TIME-OUT SET	100	100	100	100	100	

21

Appendix 8: Other Functions for Reference

21.1 F1-digit Feed

The feedrate registered by parameter can be assigned by designating a single digit following address F.
There are six F codes: F0 and F1 to F5. The rapid traverse rate is applied when F0 is issued which is the same as the G00 command.
When one of the codes from F1 to F5 is issued, the cutting feedrate set to support the code serves as the valid rate command.
If F6 or larger value is command, the value is regarded as the cutting feedrate which has been directly commanded with numerical values.
When "F1-digit speed change valid" signal is valid when F1-digit feed is issued, a feedrate which is set by a parameter can be increased or decreased by operating the manual handle.

Refer to "Programming Manual" for more details on F1-digit feed.

Validate the F1-digit feedrate

To validate the F1-digit feed, the parameter "#1079 f1digt" or "#8145 Validate F1 digit" must be ON.

When the F1-digit feed is valid, the feedrate of each command is as follows:
F0: Rapid traverse rate
F1 to F5: The feedrate configured by the corresponding parameter "#1185 spd_F1" to "#1189 spd_F5"(F1 digit feedrate F1 to F5)

Enabling conditions of the speed change with the handle

When the F1-digit command is valid, the feedrate can be increased/decreased by operating the first manual handle. (The feedrate cannot be changed by the 2nd and 3rd handles.)

Changing the feedrate by the handle is available satisfying the following conditions:

(1) The operation must be in automatic start.
(2) The operation must be in cutting feed, and F1-digit feedrate command must be issued.
(3) The F1-digit feed parameter must be ON.
(4) The F1-digit feed switch must be ON.
(5) The operation must not be in handle mode.
(6) The operation must not be in dry run.
(7) Upper limit of F1-digit feedrate ("#1506 F1_FM") and F1-digit feedrate change constant("#1507 F1_K F1") are not zero.

Conversion of the F1-digit feedrate changing with handle operation

Once the F1-digit feedrate is changed with manual handle, the changed feedrate is valid until parameter is changed or the power is turned OFF. Changed feedrate can be checked only during the operation. Note that if the parameter "#1246 set18(bit16)" (switch over of the F1-digit feedrate changing method) is set to "1", even after the power is turned OFF, operation is kept at the changed feedrate because the parameter of F1-digit feedrate has been changed. In this case, if the screen is not switched, the changed parameter values are not reflected to the display.

The increase/decrease amount per handle gradient is expressed with the following equation.

$$\Delta F = \Delta P \times \frac{FM}{K}$$

ΔP: Handle pulse (±))
FM: Upper limit of F1-digit feedrate (#1506 F1_FM)
K: F1-digit feedrate change constant (#1507 F1_K)

(Example) When the handle scale unit is to be made ±10mm/min.

If FM is made 3600mm/min, feedrate change constant (K) must be set to "360" according to the following formula.

$$\Delta F = 10 = 1 \times \frac{3600}{K}$$

Note

(1) If F1-digit feedrate changing valid signal is turned ON when F1-digit feed is commanded and changing feedrate with the manual handle is valid, it is clamped at the upper limit of F1-digit feedrate if the F1-digit feedrate is greater than the upper limit of F1-digit feedrate.

(2) When the F1-digit feedrate is 0 with the handle operation during the F1-digit feedrate command, the alarm "M01 F1-digit feedrate zero 0104" will occur. This alarm can be eliminated when the F1-digit feedrate is set in larger value than 0 with the handle operation.

(3) When K is 0, the speed is not changed.

Precautions

(1) The number of manual handle pulses is 1 pulse per scale unit regardless of the scaling factor.

(2) During a F1-digit feed command, the F1-digit feed number and F1-digit feed command signal are output as the PLC signals.

(3) When F1. to F5. (with decimal point) are assigned, the 1mm/min to 5mm/min (direct numerical value command) are established instead of the F1-digit feed command.

(4) When the commands are used with inch command, one-tenth of the feedrate set correspond to F1 to F5 serves at the assigned speed inch/min.

(5) When the commands are used with the millimeter or degree units, the feedrate set to correspond to F1 to F5 serves as the assigned speed mm (°)/mm.

(6) When both the F1-digit feedrate changing and the manual speed command are present, the manual speed command will have the priority.

(7) In the synchronous tapping command, the speed cannot be changed with the handle.

Revision History

Revision date	Manual No.	Changes
Aug. 2025	IB(NA)1501806-J IB(NA)1501807-J	Contents of "M800V/M80V Series Instruction Manual" (IB-1501618(ENG)-H) were divided as follows: "M800V/M80V Series Instruction Manual(1/2)" (IB-1501806(ENG)-J) "M800V/M80V Series Instruction Manual(2/2)" (IB-1501807(ENG)-J) Contents were revised to correspond to Mitsubishi Electric CNC M800V/M80V Series software version B4. The following chapter was added. •3.7.6 Optimal program conversion Due to adding of the chapter as above, the existing chapter numbers were changed. •3.7.7 Displaying the Finished Shape The following chapters were changed. •1.4.5 Main Menu Functions •3.2 Creating and Editing a Machining Program •3.11.1 Screen Configuration •3.11.2 Operation Method •6.6.4 Tool Life Management Set Allocation to Part Systems •9.10 Email Notification Function to Operator •9.10.1 Setting for Email Transmission Environment •10.2.1 Screen Layout •10.4.9.4 Disabling the Interference Check •10.7 Collation and Stop •12.2 Transferring a File •12.8.2 List of File Names •15 Appendix 2: Wireless LAN Function [M800VS/M80V Series] •15.5 Troubleshooting Other mistakes were corrected.

Revision date	Manual No.	Changes
Jan. 2026	IB(NA)1501806-K IB(NA)1501807-K	Contents were revised to correspond to Mitsubishi Electric CNC M800V/M80V Series software version B8. Chapters 1, 5, 7, 9 and 10 were modified. The main changes are as follows: The following chapters were added. ♦9.2.5 Shifting the Feature Coordinate Origin ♦10.6 Interference Check III The contents were revised as follows. Chapter numbers are indicated in parentheses. ♦Added a main menu on Monitor screen. (1.4.4, 1.4.5) ♦Added the explanation to change a medium to the display items of the input/output screen. (5.1, 12) ♦Modified the screen image and display items of the machining condition selection I screen. (7.2) ♦Modified the screen image, display items, menus, operation method and precautions of the high-accuracy control parameter adjustment screen. (7.3) ♦Added a menu on the machining surface selection screen. (9.2.4) ♦Added the explanation on the display items and note of the manual numerical value command screen. (9.6) Other mistakes were corrected.
May 2026	IB(NA)1501806-L IB(NA)1501807-L	Contents were revised to correspond to Mitsubishi Electric CNC M800V/M80V Series software version BB. Chapters 4, 6, 10 and 14 were modified. The main changes are as follows. Chapter numbers are indicated in parentheses. The following chapters were added. ♦14.1 Rotation Center Error Measurement Function (M System) ♦14.1.1 Screen Configuration ♦14.1.2 Measuring the Rotation Center Error Data ♦14.2 Rotation Center Error Measurement Function The following chapter title was changed: ♦14 Rotation Center Error Measurement Function (M System) / Rotation Center Error Compensation Function The contents were revised as follows. ♦Added the items of the notes. (4.3.17) ♦Corrected mistakes about tool measurement types, and modified the explanation on tool length measurement procedure. (6.3.1) ♦Added the explanation on the counter display for orbit machining. (10.2.1) ♦Moved some explanations to Chapters 14.1 and 14.1.1. (14) Other mistakes were corrected.

Global Service Network

AMERICA

MITSUBISHI ELECTRIC AUTOMATION INC. (AMERICA FA CENTER)
HQ and Central Region Service Center (Chicago)
500 CORPORATE WOODS PARKWAY, VERNON HILLS, ILLINOIS 60061, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-847-478-2650
Minneapolis, MN Service Satellite
Detroit, MI Service Satellite
Grand Rapids, MI Service Satellite
Cleveland, OH Service Satellite
St. Louis, MO Service Satellite
Indianapolis, IN Service Saltellite

South/East Region Service Center (Georgia)
1845 SATELLITE BOULEVARD STE. 450, DULUTH, GEORGIA 30097, U.S.A.
TEL +1-678-258-4529 / FAX +1-678-258-4519
Charleston, SC Service Satellite
Charlotte, NC Service Satellite
Raleigh, NC Service Satellite
Dallas, TX Service Satellite
Houston, TX Service Satellite
Hartford, CT Service Satellite
Knoxville, TN Service Satellite
Nashville, TN Service Satellite
Huntsville, AL Satellite
Pittsburg, PA Service Satellite
Tampa, FL Service Satellite
Miami, FL Satelite
Lafayette, LA Service Satellite
Allentown, PA Satelite

Western Region Service Center (California)
5900-B KATELLA AVE. - 5900-A KATELLA AVE. CYPRESS, CALIFORNIA 90630, U.S.A.
TEL: +1-714-699-2625 / FAX: +1-847-478-2650
San Francisco, CA Satelite
Seattle, WA Service Satellite
Denver, CO Service Satellite

Canada Region Service Center (Toronto)
4299 14TH AVENUE MARKHAM, ONTARIO L3R OJ2, CANADA
TEL: +1-905-475-7728 / FAX: +1-905-475-7935
Edmonton, AB Service Satellite
Montreal, QC Service Satellite

Mexico Region Service Center (Queretaro)
Parque Tecnológico Innovación Querétaro, Lateral Carretera Estatal 431, Km 2+200, Lote 91 Modulos 1 y 2
Hacienda la Machorra, CP 76246, El Marqués, Querétaro, México
TEL: +52-442-153-6050
Monterrey, NL Service Satellite
Mexico City, DF Service Satellite

BRAZIL

MITSUBISHI ELECTRIC DO BRASIL COMÉRCIO E SERVIÇOS LTDA.
Votorantim Office
AV. GISELE CONSTANTINO,1578, PARQUE BELA VISTA, VOTORANTIM-SP, BRAZIL CEP:18.110-650
TEL: +55-15-3023-9000
Blumenau, Santa Catarina Office

EUROPE

MITSUBISHI ELECTRIC EUROPE B.V.
European Service Headquarters (Dusseldorf, GERMANY)
Mitsubishi-Electric-Platz 1 40882 RATINGEN, GERMANY
TEL: +49-2102-486-5000 / FAX: +49-2102-486-5910

South Germany Service Center (Stuttgart)
SCHELMENWASENSTRASSE 16-20, 70567 STUTTGART, GERMANY
TEL: + 49-711-770598-123 / FAX: +49-711-770598-141

France Service Center (Paris)
2 RUE DE L'UNION, 92565 RUEIL-MALMAISON CEDEX, FRANCE
TEL: +33-1-41-02-83-13 / FAX: +33-1-49-01-07-25

France Service Satellite (Lyon)
240, ALLEE JACQUES MONOD 69800 SAINT PRIEST FRANCE
TEL: +33-1-41-02-83-13 / FAX: +33-1-49-01-07-25

Italy Service Center (Milan)
VIA ENERGY PARK 14, VIMERCATE 20871 (MB) ITALY
TEL: +39-039-6053-342 / FAX: +39-039-6053-206

Italy Service Satellite (Padova)
VIA G. SAVELLI, 24 - 35129 PADOVA, ITALY
TEL: +39-039-6053-342 / FAX: +39-039-6053-206

U.K. Service Center
TRAVELLERS LANE, HATFIELD, HERTFORDSHIRE, AL10 8XB, U.K.
TEL: +44-1707-288-780 / FAX: +44-1707-278-695

Spain Service Center
CTRA. RUBI, 76-80 8174 SAINT CUGAT DEL VALLES, BARCELONA, SPAIN
TEL: +34-935-65-2236 / FAX: +34-935-89-1579

Poland Service Center
UL.KRAKOWSKA 50, 32-083 BALICE, POLAND
TEL: +48-12-347-6500 / FAX: +48-12-630-4701

Hungary Service Center
MITSUBISHI ELECTRIC EUROPE B.V. HUNGARIAN BRANCH
BUDAÖRS OFFICE PARK, SZABADSÁG ÚT 117., 2040 BUDAÖRS, HUNGARY
TEL: +36-70-433-2263

Turkey Service Center
MITSUBISHI ELECTRIC TURKEY ELEKTRİK ÜRÜNLERİ A.Ş
SERIFALI MAHALLESI KALE SOKAK. NO.41 34775
UMRANIYE, ISTANBUL, TURKEY
TEL: +90-216-969-2500 / FAX: +90-216-661-44-47

Czech Republic Service Center
MITSUBISHI ELECTRIC EUROPE B.V. PRAGUE OFFICE
Pekařská 621/7, 155 00 PRAHA 5, CZECH REPUBLIC
TEL: +420-734-402-587

Sweden Service Center
HAMMARBACKEN 14, P.O.BOX 750 SE-19127, SOLLENTUNA, SWEDEN
TEL: +46-8-6251200 / FAX: +46-8-6251014

Bulgaria Service Center
AKHNATON Ltd. (Service Partner)
4 ANDREJ LJAPCHEV BLVD. POB 21, BG-1756 SOFIA, BULGARIA
TEL: +359-2-8176009 / FAX: +359-2-9744061

Ukraine Service Center (Kiev)
CSC Automation Ltd. (Service Partner)
4 B, YEVHENA SVERSTYUKA STR., 02002 KIEV, UKRAINE
TEL: +380-44-494-3344 / FAX: +380-44-494-3366

South Africa Service Center
Adroit Technologies (Service Partner)
20 WATERFORD OFFICE PARK, WATERFORD DRIVE, CNR OF WITKOPPEN ROAD,
FOURWAYS JOHANNESBURG SOUTH AFRICA
TEL: +27-11-658-8100 / FAX: +27-11-658-8101

ASEAN

MITSUBISHI ELECTRIC ASIA PTE. LTD. (ASEAN FA CENTER)
Singapore Service Center
307 ALEXANDRA ROAD MITSUBISHI ELECTRIC BUILDING SINGAPORE 159943
TEL: +65-6473-2308 / FAX: +65-6476-7439

PHILIPPINES

MELCO FACTORY AUTOMATION PHILIPPINES INC.
Head Office
128 LOPEZ RIZAL STREET, BRGY., HIGHWAY HILLS, MANDALUYONG CITY , MM PHILIPPINES 1550
TEL: +63-2-8256-8042 / FAX: +632-8637-2294

Philippines Service Center
KM.23 WEST SERVICE ROAD SSH, CUPANG ,MUNTINLUPA CITY, PHILIPPINES
TEL: +63-2-8807-0420 / FAX: +63-2-8842-5202

VIETNAM

MITSUBISHI ELECTRIC VIETNAM CO., LTD.
Vietnam Ho Chi Minh Service Center
11TH & 12TH FLOOR, VIETTEL TOWER B, 285 CACH MANG THANG 8 STREET, WARD 12, DISTRICT 10,
HO CHI MINH CITY, VIETNAM
TEL: +84-28-3910-5945 / FAX: +84-28-3910-5947

Vietnam Hanoi Service Center
14TH FLOOR, CAPITAL TOWER, 109 TRAN HUNG DAO STREET, CUA NAM WARD,
HOAN KIEM DISTRICT, HA NOI CITY, VIETNAM
TEL: +84-24-3937-8075 / FAX: +84-24-3937-8076

INDONESIA

PT. MITSUBISHI ELECTRIC INDONESIA
Indonesia Service Center (Cikarang)
JL. KENARI RAYA BLOK G2-07A, DELTA SILICON 5, LIPPO CIKARANG - BEKASI 17550, INDONESIA
TEL: +62-21-2961-7797 / FAX: +62-21-2961-7794

MALAYSIA

MITSUBISHI ELECTRIC SALES MALAYSIA SDN. BHD.
Malaysia Service Center (Kuala Lumpur Service Center)
LOT 11, JALAN 219, P.O BOX 1036, 46860 PETALING JAYA, SELANGOR DARUL EHSAN, MALAYSIA
TEL: +60-3-7626-5032

Johor Bahru Service Center
9, Jalan Perjiranan 4/6, Bandar Dato Onn, 81100 Johor.
TEL: 07-3642250 / 2256

Pulau Pinang Service Center
25-G, Pusat Perniagaan Perdana Jaya, Jalan Permatang Rawa, 14000 Bukit Mertajam, Pulau Pinang
TEL: +60-4-510-1838 / Fax: +60-4-510-1835

THAILAND

MITSUBISHI ELECTRIC FACTORY AUTOMATION (THAILAND) CO., LTD.
Thailand Service Center (Bangkok)
101, TRUE DIGITAL PARK OFFICE, 5TH FLOOR, SUKHUMVIT ROAD, BANGCHAK, PHRA KHANONG,
BANGKOK, 10260 THAILAND
TEL: +66-2-092-8600 / FAX: +66-2-043-1231-33

INDIA

MITSUBISHI ELECTRIC INDIA PVT., LTD.
Service Head office and South India Service Center (Bangalore)
CNC eXPerience Park, PLOT NO. 57/A, PEENYA PHASE 3,
PEENYA INDUSTRIAL AREA, BANGALORE 560058, KARNATAKA, INDIA
TEL : +91-80-4655-2121
Chennai Service Satellite
Coimbatore Service Satellite
Hyderabad Service Satellite

North India Service Center (Gurgaon)
PLOT 517, GROUND FLOOR, UDYOG VIHAR PHASE-III, GURUGRAM 122008, HARYANA, INDIA
TEL : +91-124-463-0300
Ludhiana Service Satellite
Panthnagar Service Satellite
Delhi Service Satellite
Jamshedpur Service Satellite
Manesar Service Satellite

West India Service Center (Pune)
ICC-Devi GAURAV TECHNOLOGY PARK, UNIT NO.402, FOURTH FLOOR, NORTH WING,
SURVEY NUMBER 191-192 (P), NEXT to INDIAN CARD CLOTHING COMPANY Ltd,
OPP. VALLABH NAGAR, PIMPRI, PUNE- 411 018, MAHARASHTRA, INDIA
TEL : +91-20-6819-2274
Kolhapur Service Satellite
Aurangabad Service Satellite
Mumbai Service Satellite

West India Service Center (Ahmedabad)
204-209, 2ND FLOOR, 31FIVE, CORPORATE ROAD PRAHLADNAGAR,
AHMEDABAD -380015, GUJARAT, INDIA
TEL : + 91-79-6777-7888
Rajkot Service Satellite

CHINA

MITSUBISHI ELECTRIC AUTOMATION (CHINA) LTD. (CHINA FA CENTER)
CNC Call Center
TEL: +86-400-921-5130

Shanghai Service Center
NO.1386 HONG QIAO ROAD, CHANG NING QU, SHANGHAI 200336, CHINA
TEL: +86-21-2322-3030 / FAX: +86-21-2322-3000*8422
Qingdao Service Center
Suzhou Service Center
Wuhan Service Center
Ningbo Service Center
Hefei Service Center
Beijing Service Center
Tianjin Service Center
Xian Service Center
Dalian Service Center
Chengdu Service Center

Shenzhen Service Center
LEVEL8, GALAXY WORLD TOWER B, 1 YABAO ROAD, LONGGANG DISTRICT,
SHENZHEN 518129, CHINA
TEL: +86-755-2399-8272 / FAX: +86-755-8229-3686
Dongguan Service Center
Xiamen Service Center

KOREA

MITSUBISHI ELECTRIC AUTOMATION KOREA CO., LTD. (KOREA FA CENTER)
Korea Seoul Service Center
8F GANGSEO HANGANG XI-TOWER A, 401 YANGCHEON-RO, GANGSEO-GU,
SEOUL 07528 KOREA
TEL: +82-2-3660-9631 / FAX: +82-2-3664-8668

Korea Daegu Service Center
2F KT BLDG. 8, HOGUK-RO, BUK-GU, DAEGU 41518, KOREA
TEL: +82-53-382-7401 / FAX: +82-53-382-7412

Korea Changwon Technical Center
#205, 888, CHANGWON-DAERO, SEONGSAN-GU, CHANGWON-SI,
GYEONGSANGNAM-DO, 51532, KOREA
TEL: +82-55-261-5559 / FAX: +82-55-261-5558

TAIWAN

MITSUBISHI ELECTRIC TAIWAN CO., LTD. (TAIWAN FA CENTER)
Taiwan Taichung Service Center
NO. 8-1, GONGYEQU 16TH RD., XITUN DIST., TAICHUNG CITY 40768 , TAIWAN
TEL: +886-4-2359-0688 / FAX: +886-4-2359-0689

Taiwan Taipei Service Center
11F, NO.88, SEC.6, ZHONGSHAN N. RD., SHILIN DIST., TAIPEI CITY 11155, TAIWAN
TEL: +886-2-2833-5430 / FAX: +886-2-2833-5433

Taiwan Tainan Service Center
11F-1, NO.30, ZHONGZHENG S. RD., YONGKANG DIST., TAINAN CITY 71067, TAIWAN
TEL: +886-6-252-5030 / FAX: +886-6-252-5031

OCEANIA

MITSUBISHI ELECTRIC AUSTRALIA PTY. LTD.
Oceania Service Center
348 VICTORIA ROAD, RYDALMERE, N.S.W. 2116 AUSTRALIA
TEL: +61-2-9684-7269/ FAX: +61-2-9684-7245

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.

COPYRIGHT 2025-2026 MITSUBISHI ELECTRIC CORPORATION ALL RIGHTS RESERVED

MITSUBISHI ELECTRIC CORPORATION
HEAD OFFICE : TOKYO BLDG.,2-7-3 MARUNOUCHI,CHIYODA-KU,TOKYO 100-8310,JAPAN

MODEL	M800V/M80V Series
MODEL CODE	100-918 100-919
Manual No.	IB-1501806

Made in JAPAN

Specifications are subject to change without notice.