MITSUBISHI ELECTRIC

Changes for the Better

Numerical Control (CNC)

Specifications Manual (Function)

M800V/M80V Series

IB-1501610(ENG)-M

Introduction

This manual describes the specifications of Mitsubishi Electric CNC.
Read this manual thoroughly and understand the product's functions and performance 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	

To safely use this CNC unit, thoroughly study the "Precautions for Safety" before use.
Be sure to keep this manual on hand so that users can refer to it at any time.
Also refer to the manuals on "Manual List" as necessary.

■ Details described in this manual

- At the beginning of the section of each specification item, a reference table for each model of the item is described.
  M: Machining center system
  L: Lathe system
  ○: Standard
  Δ: Optional
  □: Selection
- 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 this manual, related manuals and attached documents before installation, operation, programming, maintenance or inspection to ensure correct use. Understand all the conditions described in this manual before using the unit.
We rank the safety precautions into "DANGER", "WARNING" and "CAUTION" for the manuals issued by Mitsubishi Electric, including this manual.

⚠ 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 medium or minor injuries or when only property damage may occur, if handling is mistaken.**

Note that even items ranked as " ⚠ CAUTION", may lead to serious consequences depending on the situation. All the items are important and must always be observed.

The following signs indicate prohibition and compulsory.

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

The meaning of each pictorial sign is as follows.

 **CAUTION**	 **CAUTION rotated 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

Not applicable in this manual.

⚠ CAUTION

■ Items related to product and manual

⚠ **The items that are 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.**

⚠ **Some screens and functions may differ depending on each NC system (or version), and some functions may not be possible. Please confirm the specifications before starting to 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.**

⚠ **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.**

(*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.

■ Items related to start up and maintenance

⚠ **Follow the power specifications (input voltage range, frequency range, momentary power failure time range) described in this manual.**

⚠ **Follow the environment conditions (ambient temperature, humidity, vibration, atmosphere) described in this manual.**

Follow the remote type machine contact input/output interface described in this manual. (Connect a diode in parallel with the inductive load or connect a protective resistor in serial with the capacitive load, etc.)

⚠ **If the temperature rise detection function is invalidated with the parameters, the control could be disabled when the temperature is excessive. This could result in machine damage or personal injuries due to runaway axis, and could damage the device. Enable the detection function for normal use. The parameter for the temperature rise detection function will be validated forcibly when the NC unit is turned ON.**

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 .

WARRANTY

Please confirm the following product warranty details before using Mitsubishi Electric CNC.

1. Warranty Period and Coverage

Should any fault or defect (hereafter called "failure") for which we are liable occur in this product during the warranty period, repair services shall be provided at no cost through the distributor from which the product was purchased or through a Mitsubishi Electric service provider. Note, however, that this does not apply if the customer was informed prior to purchasing the product that the product is not covered under warranty. Also note that we are not responsible for any on-site readjustment and/or trial run that may be required after a defective unit is replaced.

[Warranty Term]
The term of warranty for this product shall be twenty-four (24) months from the date of delivery of the product to the end user, provided the product purchased from Mitsubishi Electric or a distributor in Japan is installed in Japan (but in no event longer than thirty (30) months, including distribution time after shipment from Mitsubishi Electric or a distributor).
Note that, in the case where the product purchased from Mitsubishi Electric or a distributor in or outside Japan is exported and installed in any country other than where it was purchased, please refer to "2. Service in Overseas Countries" below.

[Limitations]
(1) The machine tool builder is requested to conduct an initial failure diagnosis, as a general rule. The diagnosis may also be carried out by Mitsubishi Electric or our service provider for a fee at the machine tool builder's request.
(2) This warranty applies only when the conditions, method, environment, etc., of use are in compliance with the terms, conditions and instructions that are set forth in the instruction manual, user's manual, and the caution label affixed to the product, etc.
(3) Even during the term of warranty, repair costs will be charged to the customer in the following cases:
  (a) a failure caused by improper storage or handling, carelessness or negligence, etc., or a failure caused by a problem with the customer's hardware or software
  (b) a failure caused by any alteration, etc., to the product made by the customer without Mitsubishi Electric's approval
  (c) a failure which may be regarded as avoidable, if the customer's equipment in which this product is incorporated is equipped with a safety device required by applicable laws or has any function or structure considered to be indispensable in the light of common sense in the industry
  (d) a failure which could have been avoided if consumable parts designated in the instruction manual, etc. had been duly maintained and replaced
  (e) any replacement of consumable parts (including the battery, relay and fuse)
  (f) a failure caused by external factors such as inevitable accidents, including without limitation fire and abnormal fluctuation of voltage, and acts of God, including without limitation earthquakes, lightning, and natural disasters
  (g) a failure which could not have been foreseen under technologies available at the time of shipment of this product from Mitsubishi Electric
  (h) any other failures which are not attributable to Mitsubishi Electric or which the customer acknowledges are not attributable to Mitsubishi Electric

2. Service in Overseas Countries

If the customer installs a product purchased from Mitsubishi Electric in a machine or equipment and exports it to any country other than where it was purchased, the customer may sign a paid warranty contract with our local FA center.
This applies in the case where the product purchased from us in or outside Japan is exported and installed in any country other than where it was purchased.
For details please contact the distributor from which the product was purchased.

3. Exclusion of Responsibility for Compensation against Loss of Opportunity, Secondary Loss, etc.

Regardless of the gratis warranty term, Mitsubishi Electric shall not be liable for compensation for:
(1) Damage arising from any cause found not to be the responsibility of Mitsubishi Electric.
(2) Lost opportunity or lost profit incurred by the user due to a failure of a Mitsubishi Electric product.
(3) Special damage or secondary damage, whether foreseeable or not, compensation for accidents, and compensation for damages to products other than Mitsubishi Electric products.
(4) Replacement by the user, maintenance of on-site equipment, start-up test run and other tasks.

4. Changes in Product Specifications

Specifications shown in our catalogs, manuals or technical documents are subject to change without notice.

5. Product Application

(1) For use of this product, applications should be those that will not result in a serious damage even if a failure or malfunction occurs in the product, and a backup or failsafe function should operate on an external system when any failure or malfunction occurs to the product.
(2) Mitsubishi Electric CNC is designed and manufactured solely for applications to machine tools for industrial purposes. Do not use this product in applications other than those specified above, especially those which have substantial influence on public interest or which are expected to have significant influence on human lives or properties.

CONTENTS

M800V/M80V Series Specifications List [L system]

○ : Standard △ : Option □ : Selection

Class	Lathe system							Page
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB	
1 Control Axes								**1**
1.1 Control Axes								**2**
1.1.1 Number of Basic Control Axes (NC Axes)	○ 2	○ 2	○ 2	○ 2	○ 2	○ 2	○ 2	3
1.1.2 Max. Number of Axes (NC Axes + Spindles + PLC Axes)	○ 16 △ 32	○ 16 △ 32	○ 16 △ 32	○ 16 △ 32	24	24	9	3
1.1.2.1 Max. Number of NC Axes (In Total for All the Part Systems)	○ 16 △ 32	○ 16 △ 32	○ 16 △ 32	○ 16 △ 32	16	16	7	3
1.1.2.2 Max. Number of Spindles	8	8	8	8	6	6	4	3
1.1.2.3 Max. Number of PLC Axes	8	8	8	8	6	6	6	3
1.1.4 Max. Number of Indexing PLC Axes	8	8	8	8	4	4	4	4
1.1.5 Number of Simultaneous Contouring Control Axes	8	4	8	4	4	4	4	4
1.1.6 Max. Number of NC Axes in a Part System	○ 8 △ 12 △ 16	○ 8 △ 12 △ 16	○ 8 △ 12 △ 16	○ 8 △ 12 △ 16	8	8	5	4
1.1.7 Axis Name Extension	○	○	○	○	○	○	○	4
1.2 Control Part System								**5**
1.2.1 Standard Number of Part Systems	1	1	1	1	1	1	1	5
1.2.2 Max. Number of Part Systems (Main + Sub)	○ 4 △ 8	○ 4 △ 8	○ 4 △ 8	○ 4 △ 8	○ 4	○ 4	○ 2	5
1.2.2.1 Max. Number of Main Part Systems	○ 4 △ 8	○ 4 △ 8	○ 4 △ 8	○ 4 △ 8	○ 3	○ 3	○ 2	5
1.2.2.2 Max. Number of Sub Part Systems	○ 4 △ 8	○ 4 △ 8	○ 4 △ 8	○ 4 △ 8	○ 2	○ 2	○ 1	5
1.3 Control Axes and Operation Modes								**6**
1.3.1 Tape (RS-232C Input) Mode	○	○	○	○	○	○	○	6
1.3.2 Memory Mode	○	○	○	○	○	○	○	6
1.3.3 MDI Mode	○	○	○	○	○	○	○	6
1.3.4 High-Speed Program Server Mode								6
1.3.4.1 Control Unit-Side High-Speed Program Server Mode	△	△	—	—	○	—	—	6
1.3.4.2 Display Unit-Side High-Speed Program Server Mode	△	△	△	△	○	○	○	7
1.3.5 Front-Side SD Card Mode	○	○	○	○	○	○	○	7
1.3.6 Front-Side USB Memory Mode	○	○	○	○	○	○	○	8
2 Input Command								**9**
2.1 Data Increment								**10**
2.1.1 Least Command Increment								10
Least command increment 1μm	○	○	○	○	○	○	○	10
Least command increment 0.1μm	○	○	○	○	○	○	○	10
Least command increment 0.01μm(10nm)	△	△	△	△	—	—	—	10
Least command increment 0.001μm(1nm)	△	△	△	△	—	—	—	10
2.1.2 Least Control Increment								12
Least control increment 0.01μm(10nm)	○	○	○	○	○	○	○	12
Least control increment 0.001μm(1nm)	○	○	○	○	○	○	○	12
2.1.3 Indexing Increment	○	○	○	○	○	○	○	12
2.2 Unit System								**13**
2.2.1 Inch/Metric Changeover	○	○	○	○	○	○	○	13
2.2.2 Input Command Increment Tenfold	—	—	—	—	—	—	—	14

○ : Standard △ : Option □ : Selection

Class	Lathe system							Page
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB	
2.3 Program Format								**15**
2.3.1 Program Format								15
2.3.1.1 Format 1 for Lathe (G Code List 2, 3)	○	○	○	○	○	○	○	15
2.3.1.2 Format 2 for Lathe (G Code List 4, 5)	○	○	○	○	○	○	○	15
2.3.1.3 Special Format for Lathe (G Code List 6, 7)	○	○	○	○	○	○	○	15
2.3.1.4 Format 1 for Machining Center	—	—	—	—	—	—	—	15
2.3.1.5 Format 2 for Machining Center (M2 Format)	—	—	—	—	—	—	—	15
2.3.1.6 MITSUBISHI CNC Special Format	○	○	○	○	○	○	○	15
2.3.2 Program Format Switch	△	△	△	△	—	—	—	16
2.4 Command Value								**18**
2.4.1 Decimal Point Input I, II	○	○	○	○	○	○	○	18
2.4.2 Absolute/Incremental Command	○	○	○	○	○	○	○	19
2.4.3 Diameter/Radius Designation	○	○	○	○	○	○	○	20
2.4.4 Diameter/Radius Designation Switch	○	○	○	○	○	○	○	21
3 Positioning/Interpolation								**23**
3.1 Positioning								**24**
3.1.1 Positioning	○	○	○	○	○	○	○	24
3.1.2 Unidirectional Positioning	—	—	—	—	—	—	—	25
3.2 Linear/Circular Interpolation								**26**
3.2.1 Linear Interpolation	○	○	○	○	○	○	○	26
3.2.2 Circular Interpolation (Center/Radius Designation)	○	○	○	○	○	○	○	27
3.2.3 Helical Interpolation	○	○	○	○	○	○	○	29
3.2.4 Spiral/Conical Interpolation	—	—	—	—	—	—	—	32
3.2.5 Cylindrical Interpolation	△	△	△	△	○	○	○	34
3.2.6 Polar Coordinate Interpolation	△	△	△	△	○	○	○	35
3.2.7 Milling Interpolation	△	△	△	△	○	○	○	36
3.2.8 Hypothetical Axis Interpolation	—	—	—	—	—	—	—	37
3.2.9 Eccentric Turning	△	△	△	△	○	○	—	38
3.3 Curve Interpolation								**39**
3.3.1 Involute Interpolation	—	—	—	—	—	—	—	39
3.3.2 Exponential Interpolation	△	△	△	△	—	—	—	41
3.3.3 Spline Interpolation (G05.1Q2/G61.2)	—	—	—	—	—	—	—	42
3.3.4 NURBS Interpolation	—	—	—	—	—	—	—	43
3.3.5 3-Dimensional Circular Interpolation	—	—	—	—	—	—	—	44
3.3.6 Spline Interpolation 2 (G61.4)	—	—	—	—	—	—	—	45
4 Feed								**47**
4.1 Feedrate								**48**
4.1.1 Rapid Traverse Rate (m/min)	1000	1000	1000	1000	1000	1000	1000	48
4.1.2 Cutting Feedrate (m/min)	1000	1000	1000	1000	1000	1000	1000	49
4.1.3 Manual Feedrate (m/min)	1000	1000	1000	1000	1000	1000	1000	51
4.1.4 Rotary Axis Command Speed Tenfold	○	○	○	○	○	○	○	51
4.2 Feedrate Input Methods								**52**
4.2.1 Feed per Minute (Asynchronous Feed)	○	○	○	○	○	○	○	52
4.2.2 Feed per Revolution (Synchronous Feed)	○	○	○	○	○	○	○	53
4.2.3 Inverse Time Feed	—	—	—	—	—	—	—	54
4.2.4 F 1-Digit Feed	○	○	○	○	○	○	○	55
4.2.5 Manual Speed Command	△	△	△	△	○	○	○	56
4.2.7 G00 Feedrate Designation (,F Command)	△	△	△	△	○	○	○	57
4.2.8 Selection of Axis (Axes) for Feedrate Command	—	—	—	—	—	—	—	58
4.3 Override								**59**
4.3.1 Rapid Traverse Override	○	○	○	○	○	○	○	59
4.3.2 Cutting Feed Override	○	○	○	○	○	○	○	59

○ : Standard △ : Option □ : Selection

Class	Lathe system							Page
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB	
4.3.3 2nd Cutting Feed Override	○	○	○	○	○	○	○	59
4.3.4 Override Cancel	○	○	○	○	○	○	○	60
4.4 Acceleration/Deceleration								**61**
4.4.1 Automatic Acceleration/Deceleration After Interpolation	○	○	○	○	○	○	○	61
4.4.2 Rapid Traverse Constant-Gradient Acceleration/Deceleration	○	○	○	○	○	○	○	63
4.4.3 Rapid Traverse Constant-Gradient Multi-Step Acceleration/Deceleration	—	—	—	—	—	—	—	64
4.5 Thread Cutting								**66**
4.5.1 Thread Cutting (Lead/Thread Number Designation)	○	○	○	○	○	○	○	66
4.5.2 Variable Lead Thread Cutting	○	○	○	○	○	○	○	68
4.5.3 Synchronous Tapping								69
4.5.3.1 Synchronous Tapping Cycle	○	○	○	○	○	○	○	69
4.5.3.2 Pecking Tapping Cycle	△	△	△	△	○	○	○	71
4.5.3.3 Deep-Hole Tapping Cycle	△	△	△	△	○	○	○	73
4.5.3.5 Synchronous Tapping with Analog I/F Spindle	○	○	○	○	○	○	○	75
4.5.4 Chamfering	○	○	○	○	○	○	○	75
4.5.6 Circular Thread Cutting	△	△	△	△	○	○	—	76
4.5.8 High-Speed Synchronous Tapping (OMR-DD)	○	○	○	○	○	○	○	77
4.5.10 Thread Recutting	△	△	△	△	○	○	—	78
4.5.11 Thread Cutting Override	△	△	△	△	○	○	—	79
4.5.12 Variable Feed Thread Cutting	△	△	△	△	○	○	—	79
4.5.13 Thread Cutting Time Constant Switch	○	○	○	○	○	○	○	80
4.6 Manual Feed								**81**
4.6.1 Manual Rapid Traverse	○	○	○	○	○	○	○	81
4.6.2 Jog Feed	○	○	○	○	○	○	○	81
4.6.3 Incremental Feed	○	○	○	○	○	○	○	82
4.6.4 Handle Feed	○	○	○	○	○	○	○	82
4.6.5 Manual Feedrate B	○	○	○	○	○	○	○	83
4.6.6 Manual Feedrate B Surface Speed Control	—	—	—	—	—	—	—	84
4.6.8 Manual Speed Clamp	○	○	○	○	○	○	○	85
4.7 Dwell								**86**
4.7.1 Dwell (Time-Based Designation)	○	○	○	○	○	○	○	86
4.7.2 Dwell (Revolution-Based Designation)	○	○	○	○	○	○	○	87
5 Program Memory/Editing								**89**
5.1 Memory Capacity								**90**
5.1.1 Memory Capacity (Number of Programs Stored)								90
500KB[1280m] (1000 programs)	○	○	○	○	○	○	○	90
1000KB[2560m] (1000 programs)	△	△	△	△	—	—	—	90
2000KB[5120m] (1000 programs)	△	△	△	△	—	—	—	90
5.1.2 Extended Memory								90
2000KB[5120m] (1000 programs)	○	○	○	○	○	○	○	90
5.2 External Signal Display Switch								**91**
5.2.1 Program Editing	○	○	○	○	○	○	○	91
5.2.2 Background Editing	○	○	○	○	○	○	○	92
5.2.3 Buffer Correction	○	○	○	○	○	○	○	93
5.2.5 Multi-Part System Simultaneous Program Editing	○	○	○	○	○	○	○	94
5.2.6 Special Program Editing Display for Synchronization Between Part Systems	△	△	△	△	○	○	○	94
5.2.7 Finish Shape View Programming	△	△	△	△	○	○	○	95
6 Operation and Display								**97**

○ : Standard △ : Option □ : Selection

Class	Lathe system							Page
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB	
6.1 Structure of Operation/Display Panel								**98**
6.1.1 Color Display (8.4-Type LCD TFT)	—	—	—	—	—	□	□	98
6.1.2 Color Touchscreen Display (10.4-Type LCD TFT)	—	—	□	□	—	□	□	98
6.1.3 Color Touchscreen Display (15-Type LCD TFT)	—	—	□	□	—	□	□	98
6.1.4 Color Touchscreen Display (19-Type LCD TFT)	—	—	□	□	—	□	□	98
6.1.8 Separated-Type Color Touchscreen Display (15-Type LCD TFT/Windows10)	□	□	—	—	□	—	—	98
6.1.9 Separated-Type Color Touchscreen Display (19-Type LCD TFT/Windows10)	□	□	—	—	□	—	—	98
6.1.10 Separated-Type Color Touchscreen Display (19-Type Horizontal LCD TFT/Windows10)	□	□	—	—	□	—	—	98
6.2 Operation Methods and Functions								**99**
6.2.1 Operation Input	○	○	○	○	○	○	○	99
6.2.2 Absolute/Incremental Setting	○	○	○	○	○	○	○	99
6.2.5 Displayed Part System Switch	○	○	○	○	○	○	○	100
6.2.6 Menu List	○	○	○	○	○	○	○	100
6.2.7 Display Switch by Operation Mode	○	○	○	○	○	○	○	100
6.2.8 External Signal Display Switch	○	○	○	○	○	○	—	100
6.2.9 Screen Saver	○	○	○	○	○	○	○	101
6.2.10 Parameter Guidance	○	○	○	○	○	○	○	101
6.2.11 Alarm Guidance	○	○	○	○	○	○	○	101
6.2.12 Machining Program Input Mistake Check Warning	△	△	△	△	—	—	—	102
6.2.14 Screenshot Capture	—	—	○	○	—	○	○	102
6.2.15 User Selectable Menu Configuration	○	○	○	○	○	○	○	102
6.2.16 PC-NC Network Automatic Connection	○	○	—	—	○	—	—	102
6.2.17 Device Open Parameter	○	○	○	○	○	○	○	103
6.2.18 SRAM Open Parameter	○	○	○	○	○	○	○	103
6.2.19 MTB Selectable Menu Configuration	○	○	○	○	○	○	○	103
6.2.20 Remote Desktop Connection	—	—	△	△	—	○	○	104
6.2.21 VNC Server	—	—	△	△	—	○	○	105
6.2.22 Web Access (NC Screen Remote Browsing)	—	—	○	○	—	○	○	106
6.3 Display Methods and Contents								**107**
6.3.1 Status Display	○	○	○	○	○	○	○	107
6.3.2 Clock Display	○	○	○	○	○	○	○	107
6.3.3 Monitor Screen Display	○	○	○	○	○	○	○	108
6.3.4 Setup Screen Display	○	○	○	○	○	○	○	108
6.3.5 Edit Screen Display	○	○	○	○	○	○	○	109
6.3.6 Diagnosis Screen Display	○	○	○	○	○	○	○	109
6.3.7 Maintenance Screen Display	○	○	○	○	○	○	○	109
6.3.8 Home Application	○	○	—	—	○	—	—	110
6.3.9 Home Screen	○	○	○	○	○	○	○	111
6.3.10 Additional Languages								112
6.3.10.1 Japanese	○	○	○	○	○	○	○	112
6.3.10.2 English	○	○	○	○	○	○	○	112
6.3.10.3 German	○	○	○	○	○	○	○	112
6.3.10.4 Italian	○	○	○	○	○	○	○	112
6.3.10.5 French	○	○	○	○	○	○	○	112
6.3.10.6 Spanish	○	○	○	○	○	○	○	112
6.3.10.7 Chinese								113
6.3.10.7.1 Chinese (Traditional Chinese Characters)	○	○	○	○	○	○	○	113
6.3.10.7.2 Chinese (Simplified Chinese Characters)	○	○	○	○	○	○	○	113
6.3.10.8 Korean	○	○	○	○	○	○	○	113

○ : Standard △ : Option □ : Selection

Class	Lathe system							Page
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB	
6.3.10.9 Portuguese	○	○	○	○	○	○	○	113
6.3.10.10 Hungarian	○	○	○	○	○	○	○	113
6.3.10.11 Dutch	○	○	○	○	○	○	○	113
6.3.10.12 Swedish	○	○	○	○	○	○	○	113
6.3.10.13 Turkish	○	○	○	○	○	○	○	113
6.3.10.14 Polish	○	○	○	○	○	○	○	114
6.3.10.15 Russian	○	○	○	○	○	○	○	114
6.3.10.16 Czech	○	○	○	○	○	○	○	114
6.3.10.17 Indonesian	○	○	○	○	○	○	○	114
6.3.10.18 Vietnamese	○	○	○	○	○	○	○	114
6.3.11 Display Axis Name Selection	○	○	○	○	○	○	○	115
7 Input/Output Functions and Devices								**117**
7.1 Input/Output Data								**118**
7.1.1 Machining Program Input/Output	○	○	○	○	○	○	○	118
7.1.2 Tool Offset Data Input/Output	○	○	○	○	○	○	○	118
7.1.3 Common Variable Input/Output	○	○	○	○	○	○	○	118
7.1.4 Parameter Input/Output	○	○	○	○	○	○	○	118
7.1.5 History Data Output	○	○	○	○	○	○	○	118
7.1.7 System Configuration Data Output	○	○	○	○	○	○	○	118
7.2 Input/Output I/F								**119**
7.2.1 RS-232C I/F	○	○	○	○	○	○	○	119
7.2.2 SD Card I/F								119
7.2.2.1 Control Unit-Side SD Card I/F [Up to 32 GB]	○	○	—	—	○	—	—	119
7.2.2.2 Front-Side SD Card I/F [Up to 32 GB]	○	○	○	○	○	○	○	119
7.2.3 Ethernet I/F	○	○	○	○	○	○	○	119
7.2.4 Display Unit-Side Data Server I/F	○	○	○	○	○	○	○	120
7.2.5 Front-Side USB Memory I/F [Up to 32 GB]	○	○	○	○	○	○	○	120
7.2.8 Image Input I/F	—	—	□	□	—	□	□	121
7.2.9 Measuring Instrument I/F	□	□	—	—	—	—	—	122
7.3 Computer Link								**123**
7.3.1 Computer Link B	△	△	△	△	○	○	○	123
7.4 Others								**124**
7.4.1 Handy Terminal Connection	○	○	○	○	○	○	○	124
7.4.3 IP Filter Setting	○	○	○	○	○	○	○	125
8 Spindle, Tool and Miscellaneous Functions								**127**
8.1 Spindle Functions (S)								**128**
8.1.1 Spindle Control Functions								128
8.1.1.1 Spindle Digital I/F	○	○	○	○	○	○	○	129
8.1.1.2 Spindle Analog I/F	○	○	○	○	○	○	○	129
8.1.1.3 Coil Switch	○	○	○	○	○	○	○	129
8.1.1.4 Automatic Coil Switch	○	○	○	○	○	○	○	129
8.1.1.5 Encoder Input I/F	□	□	○	○	□	○	○	130
8.1.1.6 Spindle-Mode Servo Motor Control	△	△	△	△	○	○	○	131
8.1.1.7 Spindle-Mode Rotary Axis Control	—	—	—	—	—	—	—	132
8.1.1.8 Turret Gear Change Control	△	△	△	△	○	○	○	133
8.1.1.9 Spindle Control with Pulse Train Output	—	—	—	—	—	○	○	134
8.1.2 S Code Output	○	○	○	○	○	○	○	135
8.1.3 Constant Surface Speed Control	○	○	○	○	○	○	○	136
8.1.4 Spindle Override	○	○	○	○	○	○	○	138
8.1.5 Multiple-Spindle Control								138
8.1.5.1 Multiple-Spindle Control I	○	○	○	○	○	○	○	139
8.1.5.2 Multiple-Spindle Control II	○	○	○	○	○	○	○	140

○ : Standard △ : Option □ : Selection

Class	Lathe system							Page
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB	
8.1.6 Spindle Orientation	○	○	○	○	○	○	○	141
8.1.7 Spindle Position Control (Spindle/C Axis Control)								143
8.1.7.1 Spindle Position Control (Spindle/C Axis Control)	○	○	○	○	○	○	○	143
8.1.7.2 C Axis Control During Spindle Synchronization	△	△	△	△	○	○	○	144
8.1.8 Spindle Synchronization								145
8.1.8.1 Spindle Synchronization I	○	○	○	○	○	○	○	145
8.1.8.2 Spindle Synchronization II	○	○	○	○	○	○	○	146
8.1.8.3 Guide Bushing Spindle Synchronization	△	△	△	△	○	○	—	146
8.1.9 Tool Spindle Synchronization I (Polygon)								147
8.1.9.1 Tool Spindle Synchronization IA (Spindle-Spindle, Polygon)	△	△	△	△	○	○	○	147
8.1.9.2 Tool Spindle Synchronization IB (Spindle-Spindle, Polygon)	△	△	△	△	○	○	○	148
8.1.9.3 Tool Spindle Synchronization IC (Spindle-NC Axis, Polygon)	△	△	△	△	○	○	—	149
8.1.10 Tool Spindle Synchronization II (Hobbing)	△	△	△	△	○	○	—	150
8.1.11 Spindle Speed Clamp	○	○	○	○	○	○	○	151
8.1.13 Spindle Oscillation	△	△	△	△	—	—	—	151
8.1.14 Spindle Superimposition Control	△	△	△	△	○	○	—	152
8.1.15 Multiple Spindle Synchronization Set Control	○	○	○	○	○	○	○	154
8.1.16 Spindle Speed Fluctuation Detection	△	△	△	△	○	○	○	157
8.1.17 Spindle Motor Temperature Output to PLC	○	○	○	○	○	○	○	158
8.2 Tool Functions (T)								**159**
8.2.1 Tool Functions (T Command)	○	○	○	○	○	○	○	159
8.3 Miscellaneous Functions (M)								**160**
8.3.1 Miscellaneous Functions	○	○	○	○	○	○	○	160
8.3.2 Multiple M Codes in 1 Block	○	○	○	○	○	○	○	160
8.3.3 M Code Independent Output	○	○	○	○	○	○	○	161
8.3.4 Miscellaneous Function Finish	○	○	○	○	○	○	○	162
8.3.5 M Code Output during Axis Traveling	○	○	○	○	○	○	—	163
8.3.6 Miscellaneous Function Command High-speed Output	○	○	○	○	○	○	○	164
8.4 2nd Miscellaneous Functions (B)								**166**
8.4.1 2nd Miscellaneous Functions	○	○	○	○	○	○	○	166
8.4.2 2nd Miscellaneous Function Name Extension	○	○	○	○	○	○	○	166
9 Tool Compensation								**167**
9.1 Tool Length/Tool Position								**168**
9.1.1 Tool Length Offset	○	○	○	○	○	○	○	168
9.1.2 Tool Position Offset	—	—	—	—	—	—	—	171
9.1.3 Tool Compensation for Additional Axes	○	○	○	○	○	○	○	171
9.1.4 Tool Position Compensation (G43.7)	—	—	—	—	—	—	—	172
9.2 Tool Radius								**173**
9.2.1 Tool Radius Compensation	—	—	—	—	—	—	—	173
9.2.2 3-Dimensional Tool Radius Compensation	△ (*1)	△ (*1)	—	—	—	—	—	175
9.2.3 Tool Nose Radius Compensation (G40/41/42)	○	○	○	○	○	○	○	176
9.2.4 Automatic Decision of Nose Radius Compensation Direction (G46/40)	○	○	○	○	○	○	○	177
9.2.5 Tool Radius Compensation Diameter Designation	—	—	—	—	—	—	—	177
9.3 Tool Offset Amount								**178**
9.3.1 Number of Tool Offset Sets								178
99 sets	—	—	—	—	—	—	○	178
128 sets	○	○	○	○	—	—	—	178
200 sets	—	—	—	—	—	—	—	178

○ : Standard △ : Option □ : Selection

Class	Lathe system							Page
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB	
256 sets	—	—	—	—	○	○	—	178
400 sets	△	△	△	△	—	—	—	178
999 sets	△	△	△	△	—	—	—	178
9.3.2 Offset Memory								179
9.3.2.1 Tool Shape/Wear Offset Amount	○	○	○	○	○	○	○	179
9.3.2.2 Compensation Type Selection by Parameter	—	—	—	—	—	—	—	182
9.3.3 Number of Tool Offset Sets Allocation to Part Systems	○	○	○	○	○	○	○	183
10 Coordinate System								**185**
10.1 Coordinate System Type and Setting								**186**
10.1.1 Machine Coordinate System	○	○	○	○	○	○	○	187
10.1.2 Coordinate System Setting	○	○	○	○	○	○	○	188
10.1.3 Automatic Coordinate System Setting	○	○	○	○	○	○	○	189
10.1.4 Workpiece Coordinate System Selection								190
10.1.4.1 Workpiece Coordinate System Selection (6 Sets)	○	○	○	○	○	○	○	191
10.1.4.2 Extended Workpiece Coordinate System Selection (48 Sets) G54.1P1 to P48	△	△	△	△	○	○	○	192
10.1.4.3 Extended Workpiece Coordinate System Selection (96 Sets) G54.1P1 to P96	—	—	—	—	—	—	—	192
10.1.4.4 Extended Workpiece Coordinate System Selection (300 Sets) G54.1P1 to P300	—	—	—	—	—	—	—	192
10.1.5 External Workpiece Coordinate Offset	○	○	○	○	○	○	○	193
10.1.6 Workpiece Coordinate System Preset (G92.1)	○	○	○	○	○	○	○	194
10.1.7 Local Coordinate System	○	○	○	○	○	○	○	195
10.1.8 Coordinate System for Rotary Axis	○	○	○	○	○	○	○	196
10.1.9 Plane Selection	○	○	○	○	○	○	○	197
10.1.10 Origin Set/Origin Cancel	○	○	○	○	○	○	○	198
10.1.11 Counter Set	○	○	○	○	○	○	○	200
10.1.12 Workpiece Position Offset for Rotary Axis	—	—	—	—	—	—	—	201
10.1.13 Workpiece Coordinate System Shift	○	○	○	○	○	○	○	203
10.2 Return								**204**
10.2.1 Manual Reference Position Return	○	○	○	○	○	○	○	204
10.2.2 Automatic 1st Reference Position Return	○	○	○	○	○	○	○	205
10.2.3 2nd, 3rd, and 4th Reference Position Return	○	○	○	○	○	○	○	207
10.2.4 Reference Position Check	○	○	○	○	○	○	○	209
10.2.5 Absolute Position Detection	○	○	○	○	○	○	○	210
10.2.6 Tool Exchange Position Return	○	○	○	○	○	○	○	211
11 Operation Support Functions								**213**
11.1 Program Control								**214**
11.1.1 Optional Block Skip	○	○	○	○	○	○	○	214
11.1.2 Optional Block Skip Addition	○	○	○	○	○	○	○	214
11.1.3 Single Block	○	○	○	○	○	○	○	215
11.2 Program Test								**216**
11.2.1 Dry Run	○	○	○	○	○	○	○	216
11.2.2 Machine Lock	○	○	○	○	○	○	○	216
11.2.3 Miscellaneous Function Lock	○	○	○	○	○	○	○	216
11.2.4 Graphic Check								217
11.2.4.1 Graphic Check	○	○	○	○	○	○	○	217
11.2.4.2 3D Solid Program Check	○	○	○	○	○	○	○	218
11.2.4.3 Graphic Check Rotary Axis Drawing	△	△	△	△	○	○	○	219
11.2.4.4 3D Machining Simulation	—	—	—	—	—	—	—	220
11.2.5 Graphic Trace								221

○ : Standard △ : Option □ : Selection

Class	Lathe system							Page
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB	
11.2.5.1 Graphic Trace	○	○	○	○	○	○	○	221
11.2.5.2 Graphic Trace Rotary Axis Drawing	△	△	△	△	○	○	○	222
11.2.6 Machining Time Calculation	○	○	○	○	○	○	○	222
11.2.7 Manual Arbitrary Reverse Run (Program Check Operation)	△	△	△	△	○	○	○	223
11.2.8 High-Speed Simple Program Check	○	○	○	○	○	○	○	224
11.3 Program Search/Start/Stop								**225**
11.3.1 Program Search	○	○	○	○	○	○	○	225
11.3.2 Sequence Number Search	○	○	○	○	○	○	○	225
11.3.3 Collation Stop	○	○	○	○	○	○	○	226
11.3.4 Program Restart	○	○	○	○	○	○	○	227
11.3.5 Automatic Operation Start	○	○	○	○	○	○	○	228
11.3.6 NC Reset	○	○	○	○	○	○	○	228
11.3.7 Feed Hold	○	○	○	○	○	○	○	229
11.3.8 Search & Start	○	○	○	○	○	○	○	229
11.3.10 Auto-restart	○	○	○	○	○	○	○	229
11.4 Interrupt Operation								**230**
11.4.1 Manual Interruption	○	○	○	○	○	○	○	230
11.4.2 Automatic Operation Handle Interruption	○	○	○	○	○	○	○	230
11.4.3 Manual Absolute Switch	○	○	○	○	○	○	○	231
11.4.4 Thread Cutting Cycle Retract	○	○	○	○	○	○	○	232
11.4.5 Tapping Retract	○	○	○	○	○	○	○	233
11.4.6 Manual Numerical Value Command	○	○	○	○	○	○	○	234
11.4.7 Arbitrary Reverse Run	—	—	—	—	—	—	—	235
11.4.8 MDI Interruption	○	○	○	○	○	○	○	236
11.4.9 Simultaneous Operation of Manual and Automatic Modes	○	○	○	○	○	○	○	236
11.4.10 Simultaneous Operation of JOG and Handle Modes	○	○	○	○	○	○	○	236
11.4.11 Reference Position Retract	○	○	○	○	○	○	○	237
11.4.12 Tool Retract and Return	△	△	△	△	○	○	○	238
11.4.13 Skip Retract	—	—	—	—	—	—	—	239
11.4.14 PLC Interruption	○	○	○	○	○	○	○	239
12 Program Support Functions								**241**
12.1 Machining Method Support Functions								**242**
12.1.1 Program								242
12.1.1.1 Subprogram Control [Layers]	○ 10	○ 10	○ 10	○ 10	○ 10	○ 10	○ 10	242
12.1.1.2 Figure Rotation	—	—	—	—	—	—	—	245
12.1.1.3 Scaling	—	—	—	—	—	—	—	247
12.1.1.4 Axis Name Switch	○	○	○	○	○	○	○	248
12.1.2 Macro Program								249
12.1.2.1 User Macro [Layers]	○ 4	○ 4	○ 4	○ 4	○ 4	○ 4	○ 4	249
12.1.2.2 Machine Tool Builder Macro	○	○	○	○	○	○	○	253
208kB (200 programs)	—	—	—	—	○	○	○	253
256kB (600 programs)	○	○	○	○	—	—	—	253
512kB (800 programs)	△	△	△	△	○	○	○	253
1024kB (1000 programs)	△	△	△	△	—	—	—	253
12.1.2.3 Macro Interruption	○	○	○	○	○	○	○	255
12.1.2.4 Variable Commands								256
600 sets	—	—	—	—	—	—	○	256
700 sets	○	○	○	○	○	○	—	256
8000 sets	△	△	△	△	○	○	—	256
(600+100×number of part systems) sets	○	○	○	○	○	○	—	256

○ : Standard △ : Option □ : Selection

Class	Lathe system							Page
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB	
(7900+100×number of part systems) sets	△	△	△	△	○	○	—	256
12.1.2.5 Extended Common Variable	○	○	○	○	○	○	○	258
12.1.3 Fixed Cycles								260
12.1.3.1 Fixed Cycle for Drilling	○	○	○	○	○	○	○	261
12.1.3.2 Fixed Cycle for Drilling (Type II)	○	○	○	○	○	○	○	267
12.1.3.3 Special Fixed Cycle	—	—	—	—	—	—	—	268
12.1.3.4 Fixed Cycle for Turning Machining	○	○	○	○	○	○	○	272
12.1.3.5 Compound Type Fixed Cycle for Turning Machining	△	△	△	△	○	○	○	277
12.1.3.6 Compound Type Fixed Cycle for Turning Machining (Type II)	△	△	△	△	○	○	○	282
12.1.3.7 Small-Diameter Deep-Hole Drilling Cycle	△	△	△	△	—	—	—	286
12.1.3.8 Two-Dimensional Barcode Engraving Cycle	○	○	○	○	○	○	○	287
12.1.3.9 Text Engraving Cycle	○	○	○	○	○	○	○	289
12.1.4 Mirror Image								290
12.1.4.1 Mirror Image by Parameter Setting	○	○	○	○	○	○	○	290
12.1.4.2 Mirror Image by External Input	○	○	○	○	○	○	○	290
12.1.4.3 Mirror Image by G Code	—	—	—	—	—	—	—	291
12.1.4.4 Mirror Image for Facing Tool Posts	○	○	○	○	○	○	○	292
12.1.4.5 T Code Mirror Image for Facing Tool Posts	○	○	○	○	○	○	○	292
12.1.5 Coordinate System Operation								293
12.1.5.1 Coordinate Rotation by Program	△	△	△	△	○	○	—	293
12.1.5.2 Coordinate Rotation by Parameter	—	—	—	—	—	—	—	295
12.1.5.3 3-Dimensional Coordinate Conversion	△	△	△	△	○	○	—	296
12.1.6 Dimension Input								297
12.1.6.1 Corner Chamfering/Corner R	△	△	△	△	○	○	○	297
12.1.6.2 Linear Angle Command	△	△	△	△	○	○	○	304
12.1.6.3 Geometric Command	○	○	○	○	○	○	○	305
12.1.6.4 Polar Coordinate Command	—	—	—	—	—	—	—	307
12.1.7 Axis Control								308
12.1.7.1 Chopping								308
12.1.7.1.1 Chopping	△	△	△	△	○	○	○	308
12.1.7.2 Normal Line Control	—	—	—	—	—	—	—	309
12.1.7.3 Circular Cutting	—	—	—	—	—	—	—	310
12.1.7.4 Vibration Cutting Control	□	□	□	□	□	□	□	311
12.1.7.5 Orbit Machining	—	—	—	—	—	—	—	312
12.1.8 Multi-Part System Control								313
12.1.8.1 Timing Synchronization Between Part Systems	○	○	○	○	○	○	○	313
12.1.8.2 Start Point Designation Timing Synchronization	○	○	○	○	○	○	○	314
12.1.8.3 Mixed Control								317
12.1.8.3.1 Mixed Control (Cross Axis Control)	△	△	△	△	○	○	—	317
12.1.8.3.2 Arbitrary Axis Exchange Control	△	△	△	△	○	○	—	318
12.1.8.4 Control Axis Superimposition								320
12.1.8.4.1 Control Axis Superimposition	△	△	△	△	○	○	—	320
12.1.8.4.2 Arbitrary Axis Superimposition Control	△	△	△	△	—	—	—	321
12.1.8.5 Control Axis Synchronization Between Part Systems	△	△	△	△	○	○	○	322
12.1.8.6 Balance Cut	○	○	○	○	○	○	—	324
12.1.8.7 Common Memory for Part Systems	○	○	○	○	○	○	—	325
12.1.8.8 Multi-Part System Simultaneous Thread Cutting								326

○ : Standard △ : Option □ : Selection

Class	Lathe system							Page
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB	
12.1.8.8.1 Two-Part System Simultaneous Thread Cutting	△	△	△	△	○	○	—	327
12.1.8.8.2 Multi-Part System Simultaneous Thread Cutting	△	△	△	△	○	○	—	329
12.1.8.9 Multi-Part System Program Management	○	○	○	○	○	○	○	330
12.1.8.10 Synchronization Between Part Systems								331
12.1.8.10.1 Single Block Between Part Systems	△	△	△	△	○	○	○	331
12.1.8.10.2 Dwell/Miscellaneous Function Time Override	△	△	△	△	○	○	○	332
12.1.8.10.3 Synchronization Between Part Systems OFF	△	△	△	△	○	○	○	333
12.1.8.11 Sub Part System Control I	△	△	△	△	○	○	○	334
12.1.8.12 Sub Part System Control II	△	△	△	△	○	○	—	336
12.1.8.13 Loader Control Part System	—	—	—	—	—	—	—	338
12.1.9 Data Input/Output by Program								339
12.1.9.1 Parameter Input by Program	○	○	○	○	○	○	○	339
12.1.9.2 Compensation Data Input by Program	○	○	○	○	○	○	○	340
12.1.9.3 Tool/Material Shape Input by Program	○	○	○	○	○	○	○	344
12.1.9.5 API Section and Sub-Section Nos. Input/Output by Program	○	○	○	○	○	○	○	346
12.1.9.6 R-Navi Data Input by Program	—	—	—	—	—	—	—	346
12.1.10 Machining Modal								347
12.1.10.1 Tapping Mode	○	○	○	○	○	○	○	347
12.1.10.2 Cutting Mode	○	○	○	○	○	○	○	347
12.1.11 High-Speed Parts Machining								348
12.1.11.1 Rapid Traverse Block Overlap	△	△	△	△	○	○	○	348
12.2 Machining Accuracy Support Functions								**350**
12.2.1 Automatic Corner Override	○	○	○	○	○	○	○	350
12.2.2 Deceleration Check								351
12.2.2.1 Exact Stop Check Mode	○	○	○	○	○	○	○	353
12.2.2.2 Exact Stop Check	○	○	○	○	○	○	○	353
12.2.2.3 Error Detection	○	○	○	○	○	○	○	353
12.2.2.4 Programmable In-Position Check	○	○	○	○	○	○	○	354
12.2.2.5 Automatic Error Detection	△	△	△	△	—	—	—	354
12.3 High-Speed and High-Accuracy Functions [kBPM: k Block per Minute]								**355**
12.3.1 High-Speed Machining Mode								355
12.3.1.1 High-Speed Machining Mode I (G05P1) Maximum [kBPM]	△ 33.7	△ 33.7	△ 33.7	△ 33.7	○ 33.7	○ 33.7	—	355
12.3.1.2 High-Speed Machining Mode II (G05P2) Maximum [kBPM]	△ 168	△ 168	△ 168	△ 168	○ 101	○ 101	—	357
12.3.2 High-Accuracy Control								359
12.3.2.1 High-Accuracy Control (G61.1/G08)	△	△	△	△	○	○	○	359
12.3.2.2 Multi-Part System Simultaneous High-Accuracy Control	△ 3	△ 3	△ 3	△ 3	—	—	—	363
12.3.2.3 SSS Control	△	△	△	△	○	○	○	364
12.3.2.4 Tolerance Control	△	△	△	△	○	○	○	365
12.3.2.5 Variable-Acceleration Pre-Interpolation Acceleration/Deceleration	—	—	—	—	—	—	—	366
12.3.2.6 High-Accuracy Acceleration/Deceleration Time Constant Extension	—	—	—	—	—	—	—	366
12.3.2.7 Axis-Specific Acceleration Tolerance Control	△	△	△	△	—	—	—	367
12.3.3 High-Speed High-Accuracy Control								368
12.3.3.1 High-Speed High-Accuracy Control I (G05.1Q1) Maximum [kBPM]	△ 67.5	△ 67.5	△ 67.5	△ 67.5	○ 33.7	○ 33.7	—	368

○ : Standard △ : Option □ : Selection

Class	Lathe system							Page
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB	
12.3.3.2 High-Speed High-Accuracy Control II (G05P10000) Maximum [kBPM]	△ 168	△ 168	△ 168	△ 168	○ 101	○ 101	—	369
12.3.3.3 High-Speed High-Accuracy Control III (G05P20000) Maximum [kBPM]	—	—	—	—	—	—	—	373
12.3.3.4 Smooth Fairing	—	—	—	—	—	—	—	375
12.3.4 Machining Condition Selection I	○	○	○	○	○	○	○	376
12.3.5 Direct Command Mode	△	△	△	△	—	—	—	376
12.4 Programming Support Functions								**377**
12.4.1 Playback	△	△	△	△	—	—	—	377
12.4.3 Interactive Cycle Insertion	△	△	△	△	○	○	○	378
12.4.4 Simple Programming (NAVI MILL/LATHE)	△	△	△	△	○	○	○	381
12.4.5 G Code Guidance	○	○	○	○	○	○	○	382
12.4.7 DXF Data Input	△	△	△	△	○	○	—	383
12.4.8 Process Analysis	○	○	○	○	○	○	○	384
13 Machine Accuracy Compensation								**385**
13.1 Static Accuracy Compensation								**386**
13.1.1 Backlash Compensation	○	○	○	○	○	○	○	386
13.1.2 Memory-Type Pitch Error Compensation [Sets]	○ 32	○ 32	○ 32	○ 32	○ 16	○ 16	○ 16	387
13.1.3 Memory-Type Relative Position Error Compensation	○	○	○	○	○	○	○	388
13.1.4 External Machine Coordinate System Compensation	△	△	△	△	○	○	○	389
13.1.5 Circular Radius Error Compensation	△	△	△	△	○	○	○	389
13.1.6 Ball Screw Thermal Expansion Compensation	△	△	△	△	○	○	○	390
13.1.7 Rotation Center Error Compensation	△	△	△	△	—	—	—	391
13.1.8 Position-Dependent Gradually Increasing-Type Backlash Compensation	△	△	△	△	○	○	○	393
13.1.9 Bidirectional Pitch Error Compensation	△	△	△	△	○	○	○	394
13.1.10 Cyclic Error Compensation	△	△	△	△	○	○	○	394
13.1.11 Spatial Error Compensation	—	—	—	—	—	—	—	395
13.1.12 Backlash Compensation II	—	—	—	—	—	—	—	396
13.2 Dynamic Accuracy Compensation								**397**
13.2.1 Smooth High-Gain (SHG) Control	○	○	○	○	○	○	○	397
13.2.2 Dual Feedback	○	○	○	○	○	○	○	398
13.2.3 Lost Motion Compensation	○	○	○	○	○	○	○	399
13.2.4 OMR II (Backlash with Filter)	△	△	△	△	○	○	○	399
13.2.6 OMR-FF	△	△	△	△	○	○	○	400
13.2.7 Distance-Coded Reference Position Detection	△	△	△	△	○	○	○	401
13.2.8 OMR-CC (Optimum Machine Response - Contour Control)	△	△	△	△	○	○	○	402
13.2.9 Variable Torsion Compensation	—	—	—	—	—	—	—	403
13.2.10 Cogging Torque Compensation	△	△	△	△	○	○	○	404
14 Automation Support Functions								**405**
14.1 Measurement								**406**
14.1.1 Skip								406
14.1.1.1 Skip	○	○	○	○	○	○	○	406
14.1.1.2 Multiple-Step Skip	○	○	○	○	○	○	○	408
14.1.1.4 PLC Skip	△	△	△	△	○	○	○	409
14.1.1.5 Speed Change Skip	△	△	△	△	—	—	—	409
14.1.1.6 Torque Limitation Skip	△	△	△	△	○	○	○	410
14.1.2 Automatic Tool Length Measurement	○	○	○	○	○	○	○	411
14.1.3 Manual Tool Length Measurement 1	○	○	○	○	○	○	○	414
14.1.4 Manual Tool Length Measurement 2	○	○	○	○	○	○	○	417
14.1.5 Workpiece Coordinate Offset Measurement	○	○	○	○	○	○	○	418

○ : Standard △ : Option □ : Selection

Class	Lathe system							Page
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB	
14.1.6 Workpiece Position Measurement	—	—	—	—	—	—	—	419
14.1.7 Rotation Measurement	—	—	—	—	—	—	—	421
14.1.8 Rotation Center Error Measurement	—	—	—	—	—	—	—	422
14.1.9 Workpiece Installation Error Measurement	—	—	—	—	—	—	—	423
14.2 Tool Life Management								**424**
14.2.1 Tool Life Management								424
14.2.1.1 Tool Life Management I	○	○	○	○	○	○	○	424
14.2.1.2 Tool Life Management II	○	○	○	○	○	○	○	424
14.2.1.3 Tool Life Management III	—	—	—	—	—	—	—	425
14.2.2 Number of Tool Life Management Sets								426
99 sets	—	—	—	—	—	—	○	426
128 sets	○	○	○	○	—	—	—	426
200 sets	—	—	—	—	—	—	—	426
256 sets	—	—	—	—	○	○	—	426
400 sets	△	△	△	△	—	—	—	426
999 sets	△	△	△	△	—	—	—	426
14.2.3 Tool Life Management Set Allocation to Part Systems	○	○	○	○	○	○	○	427
14.3 Others								**429**
14.3.1 Programmable Current Limitation	○	○	○	○	○	○	○	429
14.3.2 Auto Power OFF	○	○	○	○	○	○	○	429
14.3.4 Load Monitoring I	△	△	△	△	○	○	—	430
14.3.5 Power ON/OFF Sequence	○	○	—	—	○	—	—	430
14.3.7 Direct Robot Control	□	□	□	□	□	□	□	431
14.3.8 Cutting Load Control	—	—	—	—	—	—	—	432
14.3.9 Chatter Suppression	△	△	△	△	○	○	—	433
15 Safety and Maintenance								**435**
15.1 Safety Switches								**436**
15.1.1 Emergency Stop	○	○	○	○	○	○	○	436
15.1.2 Data Protection Key	○	○	○	○	○	○	○	436
15.2 Display for Ensuring Safety								**437**
15.2.1 NC Warning	○	○	○	○	○	○	○	437
15.2.2 NC Alarm	○	○	○	○	○	○	○	438
15.2.3 Operation Stop Cause	○	○	○	○	○	○	○	438
15.2.4 Emergency Stop Cause	○	○	○	○	○	○	○	438
15.2.5 Thermal Detection	○	○	○	○	○	○	○	439
15.2.6 Battery Alarm/Warning	○	○	○	○	○	○	○	440
15.3 Protection								**441**
15.3.1 Stroke End (Over Travel)	○	○	○	○	○	○	○	441
15.3.2 Stored Stroke Limit								441
15.3.2.1 Stored Stroke Limit I/II	○	○	○	○	○	○	○	443
15.3.2.2 Stored Stroke Limit IB	△	△	△	△	○	○	○	445
15.3.2.3 Stored Stroke Limit IIB	△	△	△	△	○	○	○	446
15.3.2.4 Stored Stroke Limit IC	△	△	△	△	○	○	○	447
15.3.3 Stroke Check Before Travel	△	△	△	△	○	○	○	448
15.3.4 Chuck/Tailstock Barrier Check	○	○	○	○	○	○	○	450
15.3.5 Interlock	○	○	○	○	○	○	○	451
15.3.6 External Deceleration	○	○	○	○	○	○	○	451
15.3.7 Interference Check III	△	△	△	△	○	○	○	452
15.3.8 3D Machine Interference Check	—	—	—	—	—	—	—	453
15.3.9 Door Interlock								454
15.3.9.1 Door Interlock I	○	○	○	○	○	○	○	454
15.3.9.2 Door Interlock II	○	○	○	○	○	○	○	455

○ : Standard △ : Option □ : Selection

Class	Lathe system							Page
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB	
15.3.10 Parameter Lock	○	○	○	○	○	○	○	457
15.3.11 Program Protection (Edit Lock B, C)	○	○	○	○	○	○	○	457
15.3.12 Program Display Lock	○	○	○	○	○	○	○	457
15.3.13 Data Protection by User's Level	△	△	△	△	○	○	○	458
15.3.15 Vertical Axis Pull-Up	○	○	○	○	○	○	○	458
15.3.16 Machine Group-Based Alarm Stop	△	△	△	△	○	○	○	459
15.3.17 Interference Check Between Part Systems	○	○	○	○	○	○	—	461
15.3.18 Spindle Protection	○	○	○	○	○	○	○	462
15.3.19 Real-Time 3D Machine Interference Check	—	—	—	—	—	—	—	463
15.4 Maintenance and Troubleshooting								**464**
15.4.1 Operation History	○	○	○	○	○	○	○	464
15.4.2 Data Sampling	○	○	○	○	○	○	○	465
15.4.2.1 Real-time Data Sampling	○	○	○	○	○	○	○	466
15.4.3 NC Data Backup	○	○	○	○	○	○	○	466
15.4.4 Servo Tuning Support								467
15.4.4.1 NC Analyzer2	○	○	○	○	○	○	○	467
15.4.4.3 On-board Servo Tuning	○	○	○	○	○	○	○	469
15.4.5 Automatic Backup	○	○	○	○	○	○	○	470
15.4.8 Application Error Detection	○	○	—	—	○	—	—	470
15.4.9 Email Notification to Operator	△	△	△	△	○	○	○	471
15.4.10 NC Configurator2	○	○	○	○	○	○	○	472
15.4.11 Diagnosis Data Output	○	○	○	○	○	○	○	472
15.4.13 Security Feature for Windows Display	△	△	—	—	—	—	—	473
15.4.14 Field Network Diagnostics	□	□	□	□	□	□	□	474
15.4.15 Remote service iQ Care Remote4U								475
15.4.15.1 Connection with RGU	□	□	□	□	□	□	□	475
15.4.15.2 Connection with no RGU	△	△	△	△	○	○	○	475
15.4.16 Brake Wear Diagnostics	○	○	○	○	○	○	○	476
15.4.17 NC MachiningAID	○	○	○	○	○	○	○	477
15.5 Functional Safety								**478**
15.5.1 Safety Observation	△	△	△	△	○	○	○	478
15.5.2 Smart Safety Observation								479
15.5.2.1 Safety-Related I/O Observation	△	△	△	△	○	□	□	479
15.5.2.2 Emergency Stop Observation	△	△	△	△	○	□	□	479
15.5.2.3 Drive Safety Function								480
15.5.2.3.1 SLS (Safely-Limited Speed)	△	△	△	△	○	□	□	480
15.5.2.3.2 SLP (Safely-Limited Position)	△	△	△	△	○	□	□	480
15.5.2.3.3 SOS (Safe Operating Stop)	△	△	△	△	○	□	□	480
15.5.2.3.4 SSM (Safe Speed Monitor)	△	△	△	△	○	□	□	480
15.5.2.3.5 SBC/SBT (Safe Brake Control/Safe Brake Test)	△	△	△	△	○	□	□	481
15.5.2.3.6 SCA (Safe Cam)	△	△	△	△	○	□	□	481
15.5.2.3.7 SS1/SS2 (Safe Stop)	△	△	△	△	○	□	□	481
15.5.2.3.8 STO (Safe Torque Off)	△	△	△	△	○	□	□	481
16 Drive System								**483**
16.1 Servo/Spindle								**484**
16.1.1 Feed Axis								484
16.1.1.1 MDS-E-Vx	□	□	□	□	□	□	□	484
16.1.1.2 MDS-EH-Vx	□	□	□	□	□	□	□	484
16.1.1.3 MDS-EJ-Vx	□	□	□	□	□	□	□	484
16.1.1.4 MDS-EJH-Vx	□	□	□	□	□	□	□	484
16.1.1.5 MDS-EM-SPVx	□	□	□	□	□	□	□	484
16.1.1.6 MDS-EMH-SPVx	□	□	□	□	□	□	□	484

○ : Standard △ : Option □ : Selection

Class	Lathe system							Page
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB	
16.1.2 Spindle								485
16.1.2.1 MDS-E-SPx	□	□	□	□	□	□	□	485
16.1.2.2 MDS-EH-SPx	□	□	□	□	□	□	□	485
16.1.2.3 MDS-EJ-SPx	□	□	□	□	□	□	□	485
16.1.2.5 MDS-EM-SPVx	□	□	□	□	□	□	□	485
16.1.2.6 MDS-EMH-SPVx	□	□	□	□	□	□	□	485
16.1.4 Power Supply								485
16.1.4.1 MDS-E-CV	□	□	□	□	□	□	□	485
16.1.4.2 MDS-EH-CV	□	□	□	□	□	□	□	485
16.1.4.3 MDS-EX-CVP	□	□	□	□	□	□	□	485
17 Machine Support Functions								**487**
17.1 PLC								**488**
17.1.1 Built-in PLC Processing Mode	○	○	○	○	○	○	○	488
17.1.2 PLC Functions								488
17.1.2.1 Built-in PLC Basic Function	○	○	○	○	○	○	○	488
17.1.2.1.1 Index Modification	○	○	○	○	○	○	○	492
17.1.2.1.2 Multi-Program [Number of Programs]	○ 120	○ 120	○ 120	○ 120	○ 60	○ 60	○ 60	492
17.1.2.1.3 Multi-Project [Number of Projects Stored]								492
Number of PLC projects: 2	○	○	○	○	—	—	○	492
Number of PLC projects: 3	△	△	△	△	○	○	—	492
Number of PLC projects: 6	△	△	△	△	—	—	—	492
17.1.2.1.4 Function Block (FB)	○	○	○	○	○	○	○	492
17.1.2.1.5 Label Programming	○	○	○	○	○	○	○	492
17.1.2.1.6 ST Language	○	○	○	○	○	○	○	492
17.1.2.2 PLC Exclusive Instruction	○	○	○	○	○	○	○	493
17.1.2.3 Enhanced PLC Security Mode	○	○	○	○	○	○	○	494
17.1.3 PLC Support Functions								495
17.1.3.1 Alarm Message Display	○	○	○	○	○	○	○	495
17.1.3.2 Operator Message Display	○	○	○	○	○	○	○	495
17.1.3.3 Memory Switch (PLC Switch)								496
17.1.3.3.1 Memory Switch (PLC Switch) 32 Points	○	○	○	○	○	○	○	496
17.1.3.3.2 Memory Switch (PLC Switch) 64 Points	△	△	△	△	○	○	○	496
17.1.3.3.3 Memory Switch (PLC Switch) 96 Points	△	△	△	△	—	—	—	496
17.1.3.4 Load Meter Display	○	○	○	○	○	○	○	497
17.1.3.5 User PLC Version Display	○	○	○	○	○	○	○	497
17.1.3.6 Ladder Program Writing During RUN	○	○	○	○	○	○	○	497
17.1.3.7 PLC Program Protection	○	○	○	○	○	○	○	497
17.1.4 Built-in PLC Capacity								498
17.1.4.1 Standard PLC Capacity [Number of Steps]	○ 128K	○ 128K	○ 128K	○ 128K	○ 64K	○ 64K	○ 32K	498
17.1.4.2 Large PLC Capacity: 256 Ksteps	△	△	△	△	—	—	—	498
17.1.4.3 Large PLC Capacity: 512 Ksteps	△	△	△	△	—	—	—	498
17.1.5 Machine Contact Input/Output I/F	○	○	○	○	○	○	○	498
17.1.6 Ladder Monitor	○	○	○	○	○	○	○	499
17.1.7 PLC Development								500
17.1.7.1 On-Board Development	○	○	○	○	○	○	○	500
17.1.7.2 MELSEC Development Tool (GX Developer)	○	○	○	○	○	○	○	500
17.1.7.3 MELSEC Development Tool (GX Works2)	○	○	○	○	○	○	○	500
17.1.8 PLC Parameter								501
17.1.8.1 PLC Constant (150 Points)	○	○	○	○	○	○	○	501
17.1.8.2 PLC Constant Extension (Up to 750 Points)	○	○	○	○	○	○	○	501
17.1.8.3 Second PLC Constant Extension (300 Points)	○	○	○	○	○	○	○	501
17.1.9 GOT Connection								502

○ : Standard △ : Option □ : Selection

Class	Lathe system							Page
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB	
17.1.9.1 Ethernet Connection	○	○	○	○	○	○	○	502
17.1.9.2 CC-Link Connection	○	○	○	○	○	○	○	502
17.1.10 Pallet Program Registration	—	—	—	—	—	—	—	503
17.2 Machine Construction								**504**
17.2.1 Servo OFF	○	○	○	○	○	○	○	504
17.2.2 Axis Detachment	○	○	○	○	○	○	○	505
17.2.3 Synchronous Control	△	△	△	△	○	○	○	506
17.2.4 Inclined Axis Control	△	△	△	△	○	○	○	509
17.2.5 Position Switch	○ 24	○ 24	○ 24	○ 24	○ 24	○ 24	○ 24	510
17.2.7 Index Table Indexing	○	○	○	○	○	○	○	511
17.2.8 Tool Length Compensation Along the Tool Axis	△ (*1)	△ (*1)	△ (*1)	△ (*1)	—	—	—	513
17.2.9 Tool Handle Feed & Interruption	—	—	—	—	—	—	—	514
17.2.10 Tool Center Coordinate Display	—	—	—	—	—	—	—	514
17.2.11 Tool Center Point Control								515
17.2.11.1 Tool Center Point Control (G43.4/G43.5)	—	—	—	—	—	—	—	515
17.2.11.2 Tool Cutting Point Control Command (G43.8/G43.9)	—	—	—	—	—	—	—	517
17.2.12 Inclined Surface Machining Command	△	△	△	△	○	○	—	518
17.2.13 Simple Inclined Surface Machining Command	△	△	△	△	○	○	○	519
17.2.14 3-Dimensional Tool Radius Compensation (Tool's Vertical-Direction Compensation)	△ (*1)	△ (*1)	△ (*1)	△ (*1)	—	—	—	521
17.2.15 Workpiece Installation Error Compensation	△ (*1)	—	△ (*1)	—	—	—	—	522
17.2.16 3-Dimensional Manual Feed	△	△	△	△	○	○	—	523
17.2.17 R-Navi	—	—	—	—	—	—	—	524
17.2.18 Simple Synchronous Control	—	—	—	—	—	—	—	526
17.2.20 Real-Time Tuning								527
17.2.20.1 Real-Time Tuning 1 (Speed Gain)	△	△	△	△	○	○	—	527
17.2.20.2 Real-Time Tuning 2 (Rapid Traverse Time Constant)	△	△	△	△	○	○	—	528
17.2.21 Constant Torque Control	△	△	△	△	○	○	○	529
17.2.22 External Encoder Position Output I/F	○	○	○	○	○	○	○	530
17.2.23 Multiple-Axis Synchronization Control	—	—	—	—	—	—	—	531
17.2.25 BiSS Encoder I/F	—	—	○	○	—	○	○	532
17.2.26 Tool Head Hot Swapping	—	—	—	—	—	—	—	533
17.3 PLC Operation								**534**
17.3.1 Arbitrary Feed in Manual Mode	○	○	○	○	○	○	○	534
17.3.2 Circular Feed in Manual Mode	△	△	△	△	○	○	○	535
17.3.3 PLC Axis Control	○	○	○	○	○	○	○	537
17.3.5 PLC Axis Indexing	○	○	○	○	○	○	○	538
17.3.6 NC Axis/PLC Axis Switchover	△	△	△	△	○	○	○	540
17.4 PLC Interface								**541**
17.4.1 CNC Control Signal	○	○	○	○	○	○	○	541
17.4.2 CNC Status Signal	○	○	○	○	○	○	○	542
17.4.3 PLC Window	○	○	○	○	○	○	○	544
17.4.4 External Search	○	○	○	○	○	○	○	545
17.4.5 Direct Screen Selection	○	○	○	○	○	○	○	545
17.4.6 Buzzer Sound Control	○	○	○	○	○	○	○	546
17.4.7 Extended ZR Device	○	○	○	○	○	○	○	546
17.5 Machine Contact I/O								**547**
17.5.1 Operation Panel I/O								547
17.5.1.1 DI:64/DO:64	□	□	—	—	□	—	—	547
17.5.1.2 DI:64/DO:64 + SDI:8	□	□	—	—	□	—	—	547
17.5.1.3 DI:96/DO:64	—	—	□	□	—	□	□	547

○ : Standard △ : Option □ : Selection

Class	Lathe system							Page
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB	
17.5.1.5 DI:64/DO:48/AO:1	—	—	□	□	—	□	□	547
17.5.2 Remote I/O								548
17.5.2.1 DI:32/DO:32	□	□	□	□	□	□	□	548
17.5.2.2 DI:64/DO:48	□	□	□	□	□	□	□	548
17.5.2.3 DI:64/DO:48+AO:1	□	□	□	□	□	□	□	548
17.5.2.5 DI:16/DO:8	□	□	□	□	□	□	□	548
17.5.2.6 DI:32/DO:32 + SDI:8/SDO:4	□	□	□	□	□	□	□	548
17.5.2.7 SDI:8/SDO:4	□	□	□	□	□	□	□	548
17.6 External PLC Link								**549**
17.6.1 CC-Link (Master/Local)	□	□	□	□	□	□	□	549
17.6.2 PROFIBUS-DP (Master)	□	□	□	□	□	□	□	555
17.6.3 CC-Link IE Field (Master/Local)	□	□	□	□	□	□	□	556
17.6.5 EtherNet/IP	□	□	□	□	□	□	□	558
17.6.8 FL-Net	□	□	□	□	□	□	□	559
17.6.10 CC-Link IE Field Basic	○	○	○	○	○	○	○	560
17.6.11 CC-Link IE TSN (Remote Station)	□	□	□	□	□	□	□	562
17.7 Installing S/W for Machine Tools								**563**
17.7.1 Customization (NC Designer2)	△	△	△	△	○	○	○	563
17.7.1.1 Customization Data Storage Capacity [MB]	□	□	16	16	□	16	16	565
17.7.1.2 Customization Working Memory Size [MB]	6	6	6	6	6	6	6	565
17.7.2 User-Defined Key	○	○	○	○	○	○	○	565
17.7.3 EZSocket I/F	○	○	○	○	○	○	○	565
17.7.4 APLC Release	△	△	△	△	○	○	○	566
17.7.5 Custom API Library	○	○	○	○	○	○	○	567
17.7.6 MES Interface Library	△	△	△	△	○	○	○	567
17.7.7 SLMP Server	○	○	○	○	○	○	○	568
17.7.8 Numerical Control (CNC) Communication Software FCSB1224W000	○	○	○	○	○	○	○	569
17.7.11 Motion Control Release								570
17.7.11.1 Coordinate Transformation	—	—	—	—	—	—	—	570
17.7.12 Connection with High Speed Processing Unit (HPU)	□	□	—	—	□	—	—	571
17.7.13 Direct Socket Communication I/F	△	△	—	—	○	—	—	572
17.8 Others								**573**
17.8.1 System Lock	△	△	△	△	○	○	○	573
17.8.2 CNC Remote Operation Tool								573
17.8.2.1 NC Monitor2	○	○	○	○	○	○	○	573
17.8.2.2 NC Explorer	○	○	○	○	○	○	○	573
17.8.3 Automatic Operation Lock	○	○	○	○	○	○	○	574
17.8.4 Power Consumption Computation	○	○	○	○	○	○	○	574
17.8.5 EcoMonitorLight Connection	○	○	○	○	○	○	○	574
17.8.6 GOT Window	○	○	○	○	○	○	○	575
17.8.8 Laser Processing Control	—	—	—	—	—	—	—	576
17.8.9 Wireless LAN	—	—	○	○	—	○	○	578
18 Appendix 1: Functions with Advanced Machining								**579**

(*1) This function is available during program format switch.

M800V/M80V Series Specifications List [M system]

○ : Standard △ : Option □ : Selection

Class	Machining center system							Page
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB	
1 Control Axes								**1**
1.1 Control Axes								**2**
1.1.1 Number of Basic Control Axes (NC Axes)	○ 3	○ 3	○ 3	○ 3	○ 3	○ 3	○ 3	3
1.1.2 Max. Number of Axes (NC Axes + Spindles + PLC Axes)	○ 16 △ 32	○ 16 △ 32	○ 16 △ 32	○ 16 △ 32	24	24	12	3
1.1.2.1 Max. Number of NC Axes (In Total for All the Part Systems)	○ 16 △ 32	○ 16 △ 32	○ 16 △ 32	○ 16 △ 32	16	16	6	3
1.1.2.2 Max. Number of Spindles	6	6	6	6	4	4	3	3
1.1.2.3 Max. Number of PLC Axes	8	8	8	8	6	6	6	3
1.1.4 Max. Number of Indexing PLC Axes	8	8	8	8	4	4	4	4
1.1.5 Number of Simultaneous Contouring Control Axes	8	4	8	4	4	4	4	4
1.1.6 Max. Number of NC Axes in a Part System	○ 8 △ 12 △ 16	○ 8 △ 12 △ 16	○ 8 △ 12 △ 16	○ 8 △ 12 △ 16	8	8	6	4
1.1.7 Axis Name Extension	○	○	○	○	○	○	○	4
1.2 Control Part System								**5**
1.2.1 Standard Number of Part Systems	1	1	1	1	1	1	1	5
1.2.2 Max. Number of Part Systems (Main + Sub)	○ 2 △ 3(*2) △ 4 △ 8	○ 2 △ 3(*2) △ 4 △ 8	○ 2 △ 3(*2) △ 4 △ 8	○ 2 △ 3(*2) △ 4 △ 8	○ 4	○ 4	○ 2	5
1.2.2.1 Max. Number of Main Part Systems	○ 2 △ 4 △ 8	○ 2 △ 4 △ 8	○ 2 △ 4 △ 8	○ 2 △ 4 △ 8	○ 4	○ 4	○ 1	5
1.2.2.2 Max. Number of Sub Part Systems	○ 2 △ 4 △ 8	○ 2 △ 4 △ 8	○ 2 △ 4 △ 8	○ 2 △ 4 △ 8	○ 2	○ 2	○ 1	5
1.3 Control Axes and Operation Modes								**6**
1.3.1 Tape (RS-232C Input) Mode	○	○	○	○	○	○	○	6
1.3.2 Memory Mode	○	○	○	○	○	○	○	6
1.3.3 MDI Mode	○	○	○	○	○	○	○	6
1.3.4 High-Speed Program Server Mode								6
1.3.4.1 Control Unit-Side High-Speed Program Server Mode	△	△	—	—	○	—	—	6
1.3.4.2 Display Unit-Side High-Speed Program Server Mode	△	△	△	△	○	○	○	7
1.3.5 Front-Side SD Card Mode	○	○	○	○	○	○	○	7
1.3.6 Front-Side USB Memory Mode	○	○	○	○	○	○	○	8
2 Input Command								**9**
2.1 Data Increment								**10**
2.1.1 Least Command Increment								10
Least command increment 1μm	○	○	○	○	○	○	○	10
Least command increment 0.1μm	○	○	○	○	○	○	○	10
Least command increment 0.01μm(10nm)	△	△	△	△	—	—	—	10
Least command increment 0.001μm(1nm)	△	△	△	△	—	—	—	10
2.1.2 Least Control Increment								12
Least control increment 0.01μm(10nm)	○	○	○	○	○	○	○	12
Least control increment 0.001μm(1nm)	○	○	○	○	○	○	○	12
2.1.3 Indexing Increment	○	○	○	○	○	○	○	12
2.2 Unit System								**13**
2.2.1 Inch/Metric Changeover	○	○	○	○	○	○	○	13

○ : Standard △ : Option □ : Selection

Class	Machining center system							Page
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB	
2.2.2 Input Command Increment Tenfold	○	○	○	○	○	○	○	14
2.3 Program Format								**15**
2.3.1 Program Format								15
2.3.1.1 Format 1 for Lathe (G Code List 2, 3)	—	—	—	—	—	—	—	15
2.3.1.2 Format 2 for Lathe (G Code List 4, 5)	—	—	—	—	—	—	—	15
2.3.1.3 Special Format for Lathe (G Code List 6, 7)	—	—	—	—	—	—	—	15
2.3.1.4 Format 1 for Machining Center	○	○	○	○	○	○	○	15
2.3.1.5 Format 2 for Machining Center (M2 Format)	○	○	○	○	○	○	○	15
2.3.1.6 MITSUBISHI CNC Special Format	—	—	—	—	—	—	—	15
2.3.2 Program Format Switch	—	—	—	—	—	—	—	16
2.4 Command Value								**18**
2.4.1 Decimal Point Input I, II	○	○	○	○	○	○	○	18
2.4.2 Absolute/Incremental Command	○	○	○	○	○	○	○	19
2.4.3 Diameter/Radius Designation	—	—	—	—	—	—	—	20
2.4.4 Diameter/Radius Designation Switch	○	○	○	○	—	—	—	21
3 Positioning/Interpolation								**23**
3.1 Positioning								**24**
3.1.1 Positioning	○	○	○	○	○	○	○	24
3.1.2 Unidirectional Positioning	△	△	△	△	○	○	○	25
3.2 Linear/Circular Interpolation								**26**
3.2.1 Linear Interpolation	○	○	○	○	○	○	○	26
3.2.2 Circular Interpolation (Center/Radius Designation)	○	○	○	○	○	○	○	27
3.2.3 Helical Interpolation	○	○	○	○	○	○	○	29
3.2.4 Spiral/Conical Interpolation	△	△	△	△	○	○	—	32
3.2.5 Cylindrical Interpolation	△	△	△	△	○	○	○	34
3.2.6 Polar Coordinate Interpolation	△	△	△	△	—	—	—	35
3.2.7 Milling Interpolation	—	—	—	—	—	—	—	36
3.2.8 Hypothetical Axis Interpolation	△	△	△	△	—	—	—	37
3.2.9 Eccentric Turning	—	—	—	—	—	—	—	38
3.3 Curve Interpolation								**39**
3.3.1 Involute Interpolation	△	△	△	△	○	○	—	39
3.3.2 Exponential Interpolation	△	△	△	△	—	—	—	41
3.3.3 Spline Interpolation (G05.1Q2/G61.2)	△	△	△	△	○	○	—	42
3.3.4 NURBS Interpolation	△	△	△	△	—	—	—	43
3.3.5 3-Dimensional Circular Interpolation	△	△	△	△	○	○	—	44
3.3.6 Spline Interpolation 2 (G61.4)	△	△	△	△	○	○	—	45
4 Feed								**47**
4.1 Feedrate								**48**
4.1.1 Rapid Traverse Rate (m/min)	1000	1000	1000	1000	1000	1000	1000	48
4.1.2 Cutting Feedrate (m/min)	1000	1000	1000	1000	1000	1000	1000	49
4.1.3 Manual Feedrate (m/min)	1000	1000	1000	1000	1000	1000	1000	51
4.1.4 Rotary Axis Command Speed Tenfold	○	○	○	○	○	○	○	51
4.2 Feedrate Input Methods								**52**
4.2.1 Feed per Minute (Asynchronous Feed)	○	○	○	○	○	○	○	52
4.2.2 Feed per Revolution (Synchronous Feed)	△	△	△	△	○	○	○	53
4.2.3 Inverse Time Feed	△	△	△	△	○	○	—	54
4.2.4 F 1-Digit Feed	○	○	○	○	○	○	○	55
4.2.5 Manual Speed Command	△	△	△	△	○	○	○	56
4.2.7 G00 Feedrate Designation (,F Command)	△	△	△	△	○	○	○	57
4.2.8 Selection of Axis (Axes) for Feedrate Command	△	△	△	△	—	—	—	58
4.3 Override								**59**
4.3.1 Rapid Traverse Override	○	○	○	○	○	○	○	59

○ : Standard △ : Option □ : Selection

Class	Machining center system							Page
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB	
4.3.2 Cutting Feed Override	○	○	○	○	○	○	○	59
4.3.3 2nd Cutting Feed Override	○	○	○	○	○	○	○	59
4.3.4 Override Cancel	○	○	○	○	○	○	○	60
4.4 Acceleration/Deceleration								**61**
4.4.1 Automatic Acceleration/Deceleration After Interpolation	○	○	○	○	○	○	○	61
4.4.2 Rapid Traverse Constant-Gradient Acceleration/Deceleration	○	○	○	○	○	○	○	63
4.4.3 Rapid Traverse Constant-Gradient Multi-Step Acceleration/Deceleration	△	△	△	△	○	○	○	64
4.5 Thread Cutting								**66**
4.5.1 Thread Cutting (Lead/Thread Number Designation)	△	△	△	△	○	○	○	66
4.5.2 Variable Lead Thread Cutting	—	—	—	—	—	—	—	68
4.5.3 Synchronous Tapping								69
4.5.3.1 Synchronous Tapping Cycle	○	○	○	○	○	○	○	69
4.5.3.2 Pecking Tapping Cycle	△	△	△	△	○	○	○	71
4.5.3.3 Deep-Hole Tapping Cycle	△	△	△	△	○	○	○	73
4.5.3.5 Synchronous Tapping with Analog I/F Spindle	○	○	○	○	○	○	○	75
4.5.4 Chamfering	—	—	—	—	—	—	—	75
4.5.6 Circular Thread Cutting	—	—	—	—	—	—	—	76
4.5.8 High-Speed Synchronous Tapping (OMR-DD)	○	○	○	○	○	○	○	77
4.5.10 Thread Recutting	—	—	—	—	—	—	—	78
4.5.11 Thread Cutting Override	—	—	—	—	—	—	—	79
4.5.12 Variable Feed Thread Cutting	—	—	—	—	—	—	—	79
4.5.13 Thread Cutting Time Constant Switch	○	○	○	○	○	○	○	80
4.6 Manual Feed								**81**
4.6.1 Manual Rapid Traverse	○	○	○	○	○	○	○	81
4.6.2 Jog Feed	○	○	○	○	○	○	○	81
4.6.3 Incremental Feed	○	○	○	○	○	○	○	82
4.6.4 Handle Feed	○	○	○	○	○	○	○	82
4.6.5 Manual Feedrate B	○	○	○	○	○	○	○	83
4.6.6 Manual Feedrate B Surface Speed Control	△	△	△	△	—	—	—	84
4.6.8 Manual Speed Clamp	○	○	○	○	○	○	○	85
4.7 Dwell								**86**
4.7.1 Dwell (Time-Based Designation)	○	○	○	○	○	○	○	86
4.7.2 Dwell (Revolution-Based Designation)	○	○	○	○	○	○	○	87
5 Program Memory/Editing								**89**
5.1 Memory Capacity								**90**
5.1.1 Memory Capacity (Number of Programs Stored)								90
500KB[1280m] (1000 programs)	○	○	○	○	○	○	○	90
1000KB[2560m] (1000 programs)	△	△	△	△	—	—	—	90
2000KB[5120m] (1000 programs)	△	△	△	△	—	—	—	90
5.1.2 Extended Memory								90
2000KB[5120m] (1000 programs)	○	○	○	○	○	○	○	90
5.2 External Signal Display Switch								**91**
5.2.1 Program Editing	○	○	○	○	○	○	○	91
5.2.2 Background Editing	○	○	○	○	○	○	○	92
5.2.3 Buffer Correction	○	○	○	○	○	○	○	93
5.2.5 Multi-Part System Simultaneous Program Editing	○	○	○	○	—	—	—	94
5.2.6 Special Program Editing Display for Synchronization Between Part Systems	△	△	△	△	○	○	○	94
5.2.7 Finish Shape View Programming	△	△	△	△	○	○	○	95

○ : Standard △ : Option □ : Selection

Class	Machining center system							Page
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB	
6 Operation and Display								**97**
6.1 Structure of Operation/Display Panel								**98**
6.1.1 Color Display (8.4-Type LCD TFT)	—	—	—	—	—	□	□	98
6.1.2 Color Touchscreen Display (10.4-Type LCD TFT)	—	—	□	□	—	□	□	98
6.1.3 Color Touchscreen Display (15-Type LCD TFT)	—	—	□	□	—	□	□	98
6.1.4 Color Touchscreen Display (19-Type LCD TFT)	—	—	□	□	—	□	□	98
6.1.8 Separated-Type Color Touchscreen Display (15-Type LCD TFT/Windows10)	□	□	—	—	□	—	—	98
6.1.9 Separated-Type Color Touchscreen Display (19-Type LCD TFT/Windows10)	□	□	—	—	□	—	—	98
6.1.10 Separated-Type Color Touchscreen Display (19-Type Horizontal LCD TFT/Windows10)	□	□	—	—	□	—	—	98
6.2 Operation Methods and Functions								**99**
6.2.1 Operation Input	○	○	○	○	○	○	○	99
6.2.2 Absolute/Incremental Setting	○	○	○	○	○	○	○	99
6.2.5 Displayed Part System Switch	○	○	○	○	○	○	—	100
6.2.6 Menu List	○	○	○	○	○	○	○	100
6.2.7 Display Switch by Operation Mode	○	○	○	○	○	○	○	100
6.2.8 External Signal Display Switch	○	○	○	○	○	○	—	100
6.2.9 Screen Saver	○	○	○	○	○	○	○	101
6.2.10 Parameter Guidance	○	○	○	○	○	○	○	101
6.2.11 Alarm Guidance	○	○	○	○	○	○	○	101
6.2.12 Machining Program Input Mistake Check Warning	△	△	△	△	—	—	—	102
6.2.14 Screenshot Capture	—	—	○	○	—	○	○	102
6.2.15 User Selectable Menu Configuration	○	○	○	○	○	○	○	102
6.2.16 PC-NC Network Automatic Connection	○	○	—	—	○	—	—	102
6.2.17 Device Open Parameter	○	○	○	○	○	○	○	103
6.2.18 SRAM Open Parameter	○	○	○	○	○	○	○	103
6.2.19 MTB Selectable Menu Configuration	○	○	○	○	○	○	○	103
6.2.20 Remote Desktop Connection	—	—	△	△	—	○	○	104
6.2.21 VNC Server	—	—	△	△	—	○	○	105
6.2.22 Web Access (NC Screen Remote Browsing)	—	—	○	○	—	○	○	106
6.3 Display Methods and Contents								**107**
6.3.1 Status Display	○	○	○	○	○	○	○	107
6.3.2 Clock Display	○	○	○	○	○	○	○	107
6.3.3 Monitor Screen Display	○	○	○	○	○	○	○	108
6.3.4 Setup Screen Display	○	○	○	○	○	○	○	108
6.3.5 Edit Screen Display	○	○	○	○	○	○	○	109
6.3.6 Diagnosis Screen Display	○	○	○	○	○	○	○	109
6.3.7 Maintenance Screen Display	○	○	○	○	○	○	○	109
6.3.8 Home Application	○	○	—	—	○	—	—	110
6.3.9 Home Screen	○	○	○	○	○	○	○	111
6.3.10 Additional Languages								112
6.3.10.1 Japanese	○	○	○	○	○	○	○	112
6.3.10.2 English	○	○	○	○	○	○	○	112
6.3.10.3 German	○	○	○	○	○	○	○	112
6.3.10.4 Italian	○	○	○	○	○	○	○	112
6.3.10.5 French	○	○	○	○	○	○	○	112
6.3.10.6 Spanish	○	○	○	○	○	○	○	112
6.3.10.7 Chinese								113
6.3.10.7.1 Chinese (Traditional Chinese Characters)	○	○	○	○	○	○	○	113
6.3.10.7.2 Chinese (Simplified Chinese Characters)	○	○	○	○	○	○	○	113

○ : Standard △ : Option □ : Selection

Class	Machining center system							Page
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB	
6.3.10.8 Korean	○	○	○	○	○	○	○	113
6.3.10.9 Portuguese	○	○	○	○	○	○	○	113
6.3.10.10 Hungarian	○	○	○	○	○	○	○	113
6.3.10.11 Dutch	○	○	○	○	○	○	○	113
6.3.10.12 Swedish	○	○	○	○	○	○	○	113
6.3.10.13 Turkish	○	○	○	○	○	○	○	113
6.3.10.14 Polish	○	○	○	○	○	○	○	114
6.3.10.15 Russian	○	○	○	○	○	○	○	114
6.3.10.16 Czech	○	○	○	○	○	○	○	114
6.3.10.17 Indonesian	○	○	○	○	○	○	○	114
6.3.10.18 Vietnamese	○	○	○	○	○	○	○	114
6.3.11 Display Axis Name Selection	○	○	○	○	○	○	○	115
7 Input/Output Functions and Devices								**117**
7.1 Input/Output Data								**118**
7.1.1 Machining Program Input/Output	○	○	○	○	○	○	○	118
7.1.2 Tool Offset Data Input/Output	○	○	○	○	○	○	○	118
7.1.3 Common Variable Input/Output	○	○	○	○	○	○	○	118
7.1.4 Parameter Input/Output	○	○	○	○	○	○	○	118
7.1.5 History Data Output	○	○	○	○	○	○	○	118
7.1.7 System Configuration Data Output	○	○	○	○	○	○	○	118
7.2 Input/Output I/F								**119**
7.2.1 RS-232C I/F	○	○	○	○	○	○	○	119
7.2.2 SD Card I/F								119
7.2.2.1 Control Unit-Side SD Card I/F [Up to 32 GB]	○	○	—	—	○	—	—	119
7.2.2.2 Front-Side SD Card I/F [Up to 32 GB]	○	○	○	○	○	○	○	119
7.2.3 Ethernet I/F	○	○	○	○	○	○	○	119
7.2.4 Display Unit-Side Data Server I/F	○	○	○	○	○	○	○	120
7.2.5 Front-Side USB Memory I/F [Up to 32 GB]	○	○	○	○	○	○	○	120
7.2.8 Image Input I/F	—	—	□	□	—	□	□	121
7.2.9 Measuring Instrument I/F	□	□	—	—	—	—	—	122
7.3 Computer Link								**123**
7.3.1 Computer Link B	△	△	△	△	○	○	○	123
7.4 Others								**124**
7.4.1 Handy Terminal Connection	○	○	○	○	○	○	○	124
7.4.3 IP Filter Setting	○	○	○	○	○	○	○	125
8 Spindle, Tool and Miscellaneous Functions								**127**
8.1 Spindle Functions (S)								**128**
8.1.1 Spindle Control Functions								128
8.1.1.1 Spindle Digital I/F	○	○	○	○	○	○	○	129
8.1.1.2 Spindle Analog I/F	○	○	○	○	○	○	○	129
8.1.1.3 Coil Switch	○	○	○	○	○	○	○	129
8.1.1.4 Automatic Coil Switch	○	○	○	○	○	○	○	129
8.1.1.5 Encoder Input I/F	□	□	○	○	□	○	○	130
8.1.1.6 Spindle-Mode Servo Motor Control	△	△	△	△	○	○	○	131
8.1.1.7 Spindle-Mode Rotary Axis Control	△	△	△	△	○	○	—	132
8.1.1.8 Turret Gear Change Control	—	—	—	—	—	—	—	133
8.1.1.9 Spindle Control with Pulse Train Output	—	—	—	—	—	○	○	134
8.1.2 S Code Output	○	○	○	○	○	○	○	135
8.1.3 Constant Surface Speed Control	○	○	○	○	○	○	○	136
8.1.4 Spindle Override	○	○	○	○	○	○	○	138
8.1.5 Multiple-Spindle Control								138
8.1.5.1 Multiple-Spindle Control I	—	—	—	—	—	—	—	139

○ : Standard △ : Option □ : Selection

Class	Machining center system							Page
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB	
8.1.5.2 Multiple-Spindle Control II	○	○	○	○	○	○	○	140
8.1.6 Spindle Orientation	○	○	○	○	○	○	○	141
8.1.7 Spindle Position Control (Spindle/C Axis Control)								143
8.1.7.1 Spindle Position Control (Spindle/C Axis Control)	○	○	○	○	○	○	○	143
8.1.7.2 C Axis Control During Spindle Synchronization	△	△	△	△	—	—	—	144
8.1.8 Spindle Synchronization								145
8.1.8.1 Spindle Synchronization I	○	○	○	○	○	○	○	145
8.1.8.2 Spindle Synchronization II	○	○	○	○	○	○	○	146
8.1.8.3 Guide Bushing Spindle Synchronization	—	—	—	—	—	—	—	146
8.1.9 Tool Spindle Synchronization I (Polygon)								147
8.1.9.1 Tool Spindle Synchronization IA (Spindle-Spindle, Polygon)	△	△	△	△	○	○	○	147
8.1.9.2 Tool Spindle Synchronization IB (Spindle-Spindle, Polygon)	△	△	△	△	○	○	○	148
8.1.9.3 Tool Spindle Synchronization IC (Spindle-NC Axis, Polygon)	△	△	△	△	○	○	—	149
8.1.10 Tool Spindle Synchronization II (Hobbing)	△	△	△	△	○	○	—	150
8.1.11 Spindle Speed Clamp	○	○	○	○	○	○	○	151
8.1.13 Spindle Oscillation	△	△	△	△	—	—	—	151
8.1.14 Spindle Superimposition Control	—	—	—	—	—	—	—	152
8.1.15 Multiple Spindle Synchronization Set Control	—	—	—	—	—	—	—	154
8.1.16 Spindle Speed Fluctuation Detection	△	△	△	△	○	○	○	157
8.1.17 Spindle Motor Temperature Output to PLC	○	○	○	○	○	○	○	158
8.2 Tool Functions (T)								**159**
8.2.1 Tool Functions (T Command)	○	○	○	○	○	○	○	159
8.3 Miscellaneous Functions (M)								**160**
8.3.1 Miscellaneous Functions	○	○	○	○	○	○	○	160
8.3.2 Multiple M Codes in 1 Block	○	○	○	○	○	○	○	160
8.3.3 M Code Independent Output	○	○	○	○	○	○	○	161
8.3.4 Miscellaneous Function Finish	○	○	○	○	○	○	○	162
8.3.5 M Code Output during Axis Traveling	—	—	—	—	—	—	—	163
8.3.6 Miscellaneous Function Command High-speed Output	○	○	○	○	○	○	○	164
8.4 2nd Miscellaneous Functions (B)								**166**
8.4.1 2nd Miscellaneous Functions	○	○	○	○	○	○	○	166
8.4.2 2nd Miscellaneous Function Name Extension	○	○	○	○	○	○	○	166
9 Tool Compensation								**167**
9.1 Tool Length/Tool Position								**168**
9.1.1 Tool Length Offset	○	○	○	○	○	○	○	168
9.1.2 Tool Position Offset	○	○	○	○	○	○	○	171
9.1.3 Tool Compensation for Additional Axes	—	—	—	—	—	—	—	171
9.1.4 Tool Position Compensation (G43.7)	△	△	△	△	—	—	—	172
9.2 Tool Radius								**173**
9.2.1 Tool Radius Compensation	○	○	○	○	○	○	○	173
9.2.2 3-Dimensional Tool Radius Compensation	△	△	△	△	—	—	—	175
9.2.3 Tool Nose Radius Compensation (G40/41/42)	△	△	△	△	—	—	—	176
9.2.4 Automatic Decision of Nose Radius Compensation Direction (G46/40)	—	—	—	—	—	—	—	177
9.2.5 Tool Radius Compensation Diameter Designation	○	○	○	○	○	○	○	177
9.3 Tool Offset Amount								**178**
9.3.1 Number of Tool Offset Sets								178
99 sets	—	—	—	—	—	—	—	178
128 sets	—	—	—	—	—	—	—	178

○ : Standard △ : Option □ : Selection

Class	Machining center system							Page
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB	
200 sets	○	○	○	○	—	—	—	178
256 sets	—	—	—	—	—	—	—	178
400 sets	△	△	△	△	○	○	○	178
999 sets	△	△	△	△	—	—	—	178
9.3.2 Offset Memory								179
9.3.2.1 Tool Shape/Wear Offset Amount	○	○	○	○	○	○	○	179
9.3.2.2 Compensation Type Selection by Parameter	○	○	○	○	○	○	○	182
9.3.3 Number of Tool Offset Sets Allocation to Part Systems	○	○	○	○	○	○	○	183
10 Coordinate System								**185**
10.1 Coordinate System Type and Setting								**186**
10.1.1 Machine Coordinate System	○	○	○	○	○	○	○	187
10.1.2 Coordinate System Setting	○	○	○	○	○	○	○	188
10.1.3 Automatic Coordinate System Setting	○	○	○	○	○	○	○	189
10.1.4 Workpiece Coordinate System Selection								190
10.1.4.1 Workpiece Coordinate System Selection (6 Sets)	○	○	○	○	○	○	○	191
10.1.4.2 Extended Workpiece Coordinate System Selection (48 Sets) G54.1P1 to P48	△	△	△	△	○	○	○	192
10.1.4.3 Extended Workpiece Coordinate System Selection (96 Sets) G54.1P1 to P96	△	△	△	△	—	—	—	192
10.1.4.4 Extended Workpiece Coordinate System Selection (300 Sets) G54.1P1 to P300	△	△	△	△	—	—	—	192
10.1.5 External Workpiece Coordinate Offset	○	○	○	○	○	○	○	193
10.1.6 Workpiece Coordinate System Preset (G92.1)	△	△	△	△	—	—	—	194
10.1.7 Local Coordinate System	○	○	○	○	○	○	○	195
10.1.8 Coordinate System for Rotary Axis	○	○	○	○	○	○	○	196
10.1.9 Plane Selection	○	○	○	○	○	○	○	197
10.1.10 Origin Set/Origin Cancel	○	○	○	○	○	○	○	198
10.1.11 Counter Set	○	○	○	○	○	○	○	200
10.1.12 Workpiece Position Offset for Rotary Axis	△	△	△	△	○	○	—	201
10.1.13 Workpiece Coordinate System Shift	—	—	—	—	—	—	—	203
10.2 Return								**204**
10.2.1 Manual Reference Position Return	○	○	○	○	○	○	○	204
10.2.2 Automatic 1st Reference Position Return	○	○	○	○	○	○	○	205
10.2.3 2nd, 3rd, and 4th Reference Position Return	○	○	○	○	○	○	○	207
10.2.4 Reference Position Check	○	○	○	○	○	○	○	209
10.2.5 Absolute Position Detection	○	○	○	○	○	○	○	210
10.2.6 Tool Exchange Position Return	○	○	○	○	○	○	○	211
11 Operation Support Functions								**213**
11.1 Program Control								**214**
11.1.1 Optional Block Skip	○	○	○	○	○	○	○	214
11.1.2 Optional Block Skip Addition	○	○	○	○	○	○	○	214
11.1.3 Single Block	○	○	○	○	○	○	○	215
11.2 Program Test								**216**
11.2.1 Dry Run	○	○	○	○	○	○	○	216
11.2.2 Machine Lock	○	○	○	○	○	○	○	216
11.2.3 Miscellaneous Function Lock	○	○	○	○	○	○	○	216
11.2.4 Graphic Check								217
11.2.4.1 Graphic Check	○	○	○	○	○	○	○	217
11.2.4.2 3D Solid Program Check	○	○	○	○	○	○	○	218
11.2.4.3 Graphic Check Rotary Axis Drawing	—	—	—	—	—	—	—	219
11.2.4.4 3D Machining Simulation	△	△	—	—	—	—	—	220

○ : Standard △ : Option □ : Selection

Class	Machining center system							Page
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB	
11.2.5 Graphic Trace								221
11.2.5.1 Graphic Trace	○	○	○	○	○	○	○	221
11.2.5.2 Graphic Trace Rotary Axis Drawing	—	—	—	—	—	—	—	222
11.2.6 Machining Time Calculation	○	○	○	○	○	○	○	222
11.2.7 Manual Arbitrary Reverse Run (Program Check Operation)	△	△	△	△	○	○	○	223
11.2.8 High-Speed Simple Program Check	○	○	○	○	○	○	○	224
11.3 Program Search/Start/Stop								**225**
11.3.1 Program Search	○	○	○	○	○	○	○	225
11.3.2 Sequence Number Search	○	○	○	○	○	○	○	225
11.3.3 Collation Stop	○	○	○	○	○	○	○	226
11.3.4 Program Restart	○	○	○	○	○	○	○	227
11.3.5 Automatic Operation Start	○	○	○	○	○	○	○	228
11.3.6 NC Reset	○	○	○	○	○	○	○	228
11.3.7 Feed Hold	○	○	○	○	○	○	○	229
11.3.8 Search & Start	○	○	○	○	○	○	○	229
11.3.10 Auto-restart	○	○	○	○	○	○	○	229
11.4 Interrupt Operation								**230**
11.4.1 Manual Interruption	○	○	○	○	○	○	○	230
11.4.2 Automatic Operation Handle Interruption	○	○	○	○	○	○	○	230
11.4.3 Manual Absolute Switch	○	○	○	○	○	○	○	231
11.4.4 Thread Cutting Cycle Retract	○	○	○	○	○	○	—	232
11.4.5 Tapping Retract	○	○	○	○	○	○	○	233
11.4.6 Manual Numerical Value Command	○	○	○	○	○	○	○	234
11.4.7 Arbitrary Reverse Run	○	○	○	○	○	○	—	235
11.4.8 MDI Interruption	○	○	○	○	○	○	○	236
11.4.9 Simultaneous Operation of Manual and Automatic Modes	○	○	○	○	○	○	○	236
11.4.10 Simultaneous Operation of JOG and Handle Modes	○	○	○	○	○	○	○	236
11.4.11 Reference Position Retract	○	○	○	○	○	○	○	237
11.4.12 Tool Retract and Return	△	△	△	△	○	○	○	238
11.4.13 Skip Retract	○	○	○	○	○	○	○	239
11.4.14 PLC Interruption	○	○	○	○	○	○	○	239
12 Program Support Functions								**241**
12.1 Machining Method Support Functions								**242**
12.1.1 Program								242
12.1.1.1 Subprogram Control [Layers]	○ 10	○ 10	○ 10	○ 10	○ 10	○ 10	○ 10	242
12.1.1.2 Figure Rotation	△	△	△	△	—	—	—	245
12.1.1.3 Scaling	△	△	△	△	○	○	○	247
12.1.1.4 Axis Name Switch	—	—	—	—	—	—	—	248
12.1.2 Macro Program								249
12.1.2.1 User Macro [Layers]	○ 4	○ 4	○ 4	○ 4	○ 4	○ 4	○ 4	249
12.1.2.2 Machine Tool Builder Macro	○	○	○	○	○	○	○	253
208kB (200 programs)	—	—	—	—	○	○	○	253
256kB (600 programs)	○	○	○	○	—	—	—	253
512kB (800 programs)	△	△	△	△	○	○	○	253
1024kB (1000 programs)	△	△	△	△	—	—	—	253
12.1.2.3 Macro Interruption	○	○	○	○	○	○	○	255
12.1.2.4 Variable Commands								256
600 sets	—	—	—	—	—	—	—	256
700 sets	○	○	○	○	○	○	○	256
8000 sets	△	△	△	△	○	○	—	256

○ : Standard △ : Option □ : Selection

Class	Machining center system							Page
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB	
(600+100×number of part systems) sets	○	○	○	○	○	○	—	256
(7900+100×number of part systems) sets	△	△	△	△	○	○	—	256
12.1.2.5 Extended Common Variable	○	○	○	○	○	○	○	258
12.1.3 Fixed Cycles								260
12.1.3.1 Fixed Cycle for Drilling	○	○	○	○	○	○	○	261
12.1.3.2 Fixed Cycle for Drilling (Type II)	—	—	—	—	—	—	—	267
12.1.3.3 Special Fixed Cycle	△	△	△	△	○	○	○	268
12.1.3.4 Fixed Cycle for Turning Machining	○	○	○	○	○	○	—	272
12.1.3.5 Compound Type Fixed Cycle for Turning Machining	—	—	—	—	—	—	—	277
12.1.3.6 Compound Type Fixed Cycle for Turning Machining (Type II)	—	—	—	—	—	—	—	282
12.1.3.7 Small-Diameter Deep-Hole Drilling Cycle	○	○	○	○	○	○	○	286
12.1.3.8 Two-Dimensional Barcode Engraving Cycle	○	○	○	○	○	○	○	287
12.1.3.9 Text Engraving Cycle	○	○	○	○	○	○	○	289
12.1.4 Mirror Image								290
12.1.4.1 Mirror Image by Parameter Setting	○	○	○	○	○	○	○	290
12.1.4.2 Mirror Image by External Input	○	○	○	○	○	○	○	290
12.1.4.3 Mirror Image by G Code	○	○	○	○	○	○	○	291
12.1.4.4 Mirror Image for Facing Tool Posts	—	—	—	—	—	—	—	292
12.1.4.5 T Code Mirror Image for Facing Tool Posts	—	—	—	—	—	—	—	292
12.1.5 Coordinate System Operation								293
12.1.5.1 Coordinate Rotation by Program	△	△	△	△	○	○	○	293
12.1.5.2 Coordinate Rotation by Parameter	△	△	△	△	—	—	—	295
12.1.5.3 3-Dimensional Coordinate Conversion	△	△	△	△	○	○	—	296
12.1.6 Dimension Input								297
12.1.6.1 Corner Chamfering/Corner R	△	△	△	△	○	○	○	297
12.1.6.2 Linear Angle Command	△	△	△	△	○	○	○	304
12.1.6.3 Geometric Command	△	△	△	△	○	○	○	305
12.1.6.4 Polar Coordinate Command	△	△	△	△	○	○	○	307
12.1.7 Axis Control								308
12.1.7.1 Chopping								308
12.1.7.1.1 Chopping	△	△	△	△	○	○	○	308
12.1.7.2 Normal Line Control	○	○	○	○	○	○	—	309
12.1.7.3 Circular Cutting	○	○	○	○	○	○	○	310
12.1.7.4 Vibration Cutting Control	□	□	□	□	□	□	□	311
12.1.7.5 Orbit Machining	△	△	△	△	○	○	—	312
12.1.8 Multi-Part System Control								313
12.1.8.1 Timing Synchronization Between Part Systems	○	○	○	○	○	○	○	313
12.1.8.2 Start Point Designation Timing Synchronization	○	○	○	○	○	○	—	314
12.1.8.3 Mixed Control								317
12.1.8.3.1 Mixed Control (Cross Axis Control)	—	—	—	—	—	—	—	317
12.1.8.3.2 Arbitrary Axis Exchange Control	△	△	△	△	○	○	○	318
12.1.8.4 Control Axis Superimposition								320
12.1.8.4.1 Control Axis Superimposition	—	—	—	—	—	—	—	320
12.1.8.4.2 Arbitrary Axis Superimposition Control	—	—	—	—	—	—	—	321
12.1.8.5 Control Axis Synchronization Between Part Systems	△	△	△	△	○	○	—	322
12.1.8.6 Balance Cut	—	—	—	—	—	—	—	324
12.1.8.7 Common Memory for Part Systems	—	—	—	—	—	—	—	325
12.1.8.8 Multi-Part System Simultaneous Thread Cutting								326

○ : Standard △ : Option □ : Selection

Class	Machining center system							Page
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB	
12.1.8.8.1 Two-Part System Simultaneous Thread Cutting	—	—	—	—	—	—	—	327
12.1.8.8.2 Multi-Part System Simultaneous Thread Cutting	—	—	—	—	—	—	—	329
12.1.8.9 Multi-Part System Program Management	○	○	○	○	○	○	—	330
12.1.8.10 Synchronization Between Part Systems								331
12.1.8.10.1 Single Block Between Part Systems	—	—	—	—	—	—	—	331
12.1.8.10.2 Dwell/Miscellaneous Function Time Override	—	—	—	—	—	—	—	332
12.1.8.10.3 Synchronization Between Part Systems OFF	—	—	—	—	—	—	—	333
12.1.8.11 Sub Part System Control I	△	△	△	△	○	○	○	334
12.1.8.12 Sub Part System Control II	—	—	—	—	—	—	—	336
12.1.8.13 Loader Control Part System	○	○	○	○	—	—	—	338
12.1.9 Data Input/Output by Program								339
12.1.9.1 Parameter Input by Program	○	○	○	○	○	○	○	339
12.1.9.2 Compensation Data Input by Program	○	○	○	○	○	○	○	340
12.1.9.3 Tool/Material Shape Input by Program	○	○	○	○	○	○	○	344
12.1.9.5 API Section and Sub-Section Nos. Input/Output by Program	○	○	○	○	○	○	○	346
12.1.9.6 R-Navi Data Input by Program	△	△	△	△	○	○	—	346
12.1.10 Machining Modal								347
12.1.10.1 Tapping Mode	○	○	○	○	○	○	○	347
12.1.10.2 Cutting Mode	○	○	○	○	○	○	○	347
12.1.11 High-Speed Parts Machining								348
12.1.11.1 Rapid Traverse Block Overlap	△	△	△	△	○	○	○	348
12.2 Machining Accuracy Support Functions								**350**
12.2.1 Automatic Corner Override	○	○	○	○	○	○	○	350
12.2.2 Deceleration Check								351
12.2.2.1 Exact Stop Check Mode	○	○	○	○	○	○	○	353
12.2.2.2 Exact Stop Check	○	○	○	○	○	○	○	353
12.2.2.3 Error Detection	○	○	○	○	○	○	○	353
12.2.2.4 Programmable In-Position Check	○	○	○	○	○	○	○	354
12.2.2.5 Automatic Error Detection	—	—	—	—	—	—	—	354
12.3 High-Speed and High-Accuracy Functions [kBPM: k Block per Minute]								**355**
12.3.1 High-Speed Machining Mode								355
12.3.1.1 High-Speed Machining Mode I (G05P1) Maximum [kBPM]	△ 33.7	△ 33.7	△ 33.7	△ 33.7	○ 33.7	○ 33.7	○ 16.8	355
12.3.1.2 High-Speed Machining Mode II (G05P2) Maximum [kBPM]	△ 168	△ 168	△ 168	△ 168	○ 101	○ 101	○ 67.5	357
12.3.2 High-Accuracy Control								359
12.3.2.1 High-Accuracy Control (G61.1/G08)	△	△	△	△	○	○	○	359
12.3.2.2 Multi-Part System Simultaneous High-Accuracy Control	△ 3	△ 3	△ 3	△ 3	○ 3	○ 3	—	363
12.3.2.3 SSS Control	△	△	△	△	○	○	○	364
12.3.2.4 Tolerance Control	△	△	△	△	○	○	○	365
12.3.2.5 Variable-Acceleration Pre-Interpolation Acceleration/Deceleration	△	△	△	△	—	—	—	366
12.3.2.6 High-Accuracy Acceleration/Deceleration Time Constant Extension	△	△	△	△	—	—	—	366
12.3.2.7 Axis-Specific Acceleration Tolerance Control	△	△	△	△	—	—	—	367
12.3.3 High-Speed High-Accuracy Control								368
12.3.3.1 High-Speed High-Accuracy Control I (G05.1Q1) Maximum [kBPM]	△ 67.5	△ 67.5	△ 67.5	△ 67.5	○ 33.7	○ 33.7	○ 33.7	368

○ : Standard △ : Option □ : Selection

Class	Machining center system							Page
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB	
12.3.3.2 High-Speed High-Accuracy Control II (G05P10000) Maximum [kBPM]	△ 168	△ 168	△ 168	△ 168	○ 101	○ 101	○ 67.5	369
12.3.3.3 High-Speed High-Accuracy Control III (G05P20000) Maximum [kBPM]	△ 540	△ 540	△ 540	△ 540	○ 202	○ 202	—	373
12.3.3.4 Smooth Fairing	△	△	△	△	○	○	—	375
12.3.4 Machining Condition Selection I	○	○	○	○	○	○	○	376
12.3.5 Direct Command Mode	—	—	—	—	—	—	—	376
12.4 Programming Support Functions								**377**
12.4.1 Playback	○	○	○	○	○	○	○	377
12.4.3 Interactive Cycle Insertion	△	△	△	△	○	○	○	378
12.4.4 Simple Programming (NAVI MILL/LATHE)	△	△	△	△	○	○	○	381
12.4.5 G Code Guidance	○	○	○	○	○	○	○	382
12.4.7 DXF Data Input	△	△	△	△	○	○	—	383
12.4.8 Process Analysis	○	○	○	○	○	○	○	384
13 Machine Accuracy Compensation								**385**
13.1 Static Accuracy Compensation								**386**
13.1.1 Backlash Compensation	○	○	○	○	○	○	○	386
13.1.2 Memory-Type Pitch Error Compensation [Sets]	○ 32	○ 32	○ 32	○ 32	○ 16	○ 16	○ 16	387
13.1.3 Memory-Type Relative Position Error Compensation	○	○	○	○	○	○	○	388
13.1.4 External Machine Coordinate System Compensation	△	△	△	△	○	○	○	389
13.1.5 Circular Radius Error Compensation	△	△	△	△	○	○	○	389
13.1.6 Ball Screw Thermal Expansion Compensation	△	△	△	△	○	○	○	390
13.1.7 Rotation Center Error Compensation	△	△	△	△	○	○	—	391
13.1.8 Position-Dependent Gradually Increasing-Type Backlash Compensation	△	△	△	△	○	○	○	393
13.1.9 Bidirectional Pitch Error Compensation	△	△	△	△	○	○	○	394
13.1.10 Cyclic Error Compensation	△	△	△	△	○	○	○	394
13.1.11 Spatial Error Compensation	△	△	△	△	○	○	—	395
13.1.12 Backlash Compensation II	△	△	△	△	○	○	○	396
13.2 Dynamic Accuracy Compensation								**397**
13.2.1 Smooth High-Gain (SHG) Control	○	○	○	○	○	○	○	397
13.2.2 Dual Feedback	○	○	○	○	○	○	○	398
13.2.3 Lost Motion Compensation	○	○	○	○	○	○	○	399
13.2.4 OMR II (Backlash with Filter)	△	△	△	△	○	○	○	399
13.2.6 OMR-FF	△	△	△	△	○	○	○	400
13.2.7 Distance-Coded Reference Position Detection	△	△	△	△	○	○	○	401
13.2.8 OMR-CC (Optimum Machine Response - Contour Control)	△	△	△	△	○	○	○	402
13.2.9 Variable Torsion Compensation	△	△	△	△	○	○	—	403
13.2.10 Cogging Torque Compensation	△	△	△	△	○	○	○	404
14 Automation Support Functions								**405**
14.1 Measurement								**406**
14.1.1 Skip								406
14.1.1.1 Skip	○	○	○	○	○	○	○	406
14.1.1.2 Multiple-Step Skip	○	○	○	○	○	○	○	408
14.1.1.4 PLC Skip	△	△	△	△	○	○	○	409
14.1.1.5 Speed Change Skip	△	△	△	△	—	—	—	409
14.1.1.6 Torque Limitation Skip	△	△	△	△	○	○	○	410
14.1.2 Automatic Tool Length Measurement	○	○	○	○	○	○	○	411
14.1.3 Manual Tool Length Measurement 1	○	○	○	○	○	○	○	414
14.1.4 Manual Tool Length Measurement 2	○	○	○	○	○	○	○	417
14.1.5 Workpiece Coordinate Offset Measurement	—	—	—	—	—	—	—	418

○ : Standard △ : Option □ : Selection

Class	Machining center system							Page
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB	
14.1.6 Workpiece Position Measurement	○	○	○	○	○	○	○	419
14.1.7 Rotation Measurement	○	○	○	○	○	○	○	421
14.1.8 Rotation Center Error Measurement	△	△	△	△	○	○	—	422
14.1.9 Workpiece Installation Error Measurement	△	—	△	—	—	—	—	423
14.2 Tool Life Management								**424**
14.2.1 Tool Life Management								424
14.2.1.1 Tool Life Management I	○	○	○	○	○	○	○	424
14.2.1.2 Tool Life Management II	○	○	○	○	○	○	○	424
14.2.1.3 Tool Life Management III	○	○	○	○	○	○	○	425
14.2.2 Number of Tool Life Management Sets								426
99 sets	—	—	—	—	—	—	—	426
128 sets	—	—	—	—	—	—	—	426
200 sets	○	○	○	○	○	○	○	426
256 sets	—	—	—	—	—	—	—	426
400 sets	△	△	△	△	—	—	—	426
999 sets	△	△	△	△	—	—	—	426
14.2.3 Tool Life Management Set Allocation to Part Systems	○	○	○	○	○	○	○	427
14.3 Others								**429**
14.3.1 Programmable Current Limitation	○	○	○	○	○	○	○	429
14.3.2 Auto Power OFF	○	○	○	○	○	○	○	429
14.3.4 Load Monitoring I	△	△	△	△	○	○	—	430
14.3.5 Power ON/OFF Sequence	○	○	—	—	○	—	—	430
14.3.7 Direct Robot Control	□	□	□	□	□	□	□	431
14.3.8 Cutting Load Control	△	△	△	△	○	○	—	432
14.3.9 Chatter Suppression	—	—	—	—	—	—	—	433
15 Safety and Maintenance								**435**
15.1 Safety Switches								**436**
15.1.1 Emergency Stop	○	○	○	○	○	○	○	436
15.1.2 Data Protection Key	○	○	○	○	○	○	○	436
15.2 Display for Ensuring Safety								**437**
15.2.1 NC Warning	○	○	○	○	○	○	○	437
15.2.2 NC Alarm	○	○	○	○	○	○	○	438
15.2.3 Operation Stop Cause	○	○	○	○	○	○	○	438
15.2.4 Emergency Stop Cause	○	○	○	○	○	○	○	438
15.2.5 Thermal Detection	○	○	○	○	○	○	○	439
15.2.6 Battery Alarm/Warning	○	○	○	○	○	○	○	440
15.3 Protection								**441**
15.3.1 Stroke End (Over Travel)	○	○	○	○	○	○	○	441
15.3.2 Stored Stroke Limit								441
15.3.2.1 Stored Stroke Limit I/II	○	○	○	○	○	○	○	443
15.3.2.2 Stored Stroke Limit IB	△	△	△	△	○	○	○	445
15.3.2.3 Stored Stroke Limit IIB	△	△	△	△	○	○	○	446
15.3.2.4 Stored Stroke Limit IC	△	△	△	△	○	○	○	447
15.3.3 Stroke Check Before Travel	△	△	△	△	○	○	○	448
15.3.4 Chuck/Tailstock Barrier Check	—	—	—	—	—	—	—	450
15.3.5 Interlock	○	○	○	○	○	○	○	451
15.3.6 External Deceleration	○	○	○	○	○	○	○	451
15.3.7 Interference Check III	△	△	△	△	○	○	○	452
15.3.8 3D Machine Interference Check	△	△	—	—	○	—	—	453
15.3.9 Door Interlock								454
15.3.9.1 Door Interlock I	○	○	○	○	○	○	○	454
15.3.9.2 Door Interlock II	○	○	○	○	○	○	○	455

○ : Standard △ : Option □ : Selection

Class	Machining center system							Page
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB	
15.3.10 Parameter Lock	○	○	○	○	○	○	○	457
15.3.11 Program Protection (Edit Lock B, C)	○	○	○	○	○	○	○	457
15.3.12 Program Display Lock	○	○	○	○	○	○	○	457
15.3.13 Data Protection by User's Level	△	△	△	△	○	○	○	458
15.3.15 Vertical Axis Pull-Up	○	○	○	○	○	○	○	458
15.3.16 Machine Group-Based Alarm Stop	△	△	△	△	○	○	○	459
15.3.17 Interference Check Between Part Systems	—	—	—	—	—	—	—	461
15.3.18 Spindle Protection	○	○	○	○	○	○	○	462
15.3.19 Real-Time 3D Machine Interference Check	△	△	—	—	—	—	—	463
15.4 Maintenance and Troubleshooting								**464**
15.4.1 Operation History	○	○	○	○	○	○	○	464
15.4.2 Data Sampling	○	○	○	○	○	○	○	465
15.4.2.1 Real-time Data Sampling	○	○	○	○	○	○	○	466
15.4.3 NC Data Backup	○	○	○	○	○	○	○	466
15.4.4 Servo Tuning Support								467
15.4.4.1 NC Analyzer2	○	○	○	○	○	○	○	467
15.4.4.3 On-board Servo Tuning	○	○	○	○	○	○	○	469
15.4.5 Automatic Backup	○	○	○	○	○	○	○	470
15.4.8 Application Error Detection	○	○	—	—	○	—	—	470
15.4.9 Email Notification to Operator	△	△	△	△	○	○	○	471
15.4.10 NC Configurator2	○	○	○	○	○	○	○	472
15.4.11 Diagnosis Data Output	○	○	○	○	○	○	○	472
15.4.13 Security Feature for Windows Display	△	△	—	—	—	—	—	473
15.4.14 Field Network Diagnostics	□	□	□	□	□	□	□	474
15.4.15 Remote service iQ Care Remote4U								475
15.4.15.1 Connection with RGU	□	□	□	□	□	□	□	475
15.4.15.2 Connection with no RGU	△	△	△	△	○	○	○	475
15.4.16 Brake Wear Diagnostics	○	○	○	○	○	○	○	476
15.4.17 NC MachiningAID	○	○	○	○	○	○	○	477
15.5 Functional Safety								**478**
15.5.1 Safety Observation	△	△	△	△	○	○	○	478
15.5.2 Smart Safety Observation								479
15.5.2.1 Safety-Related I/O Observation	△	△	△	△	○	□	□	479
15.5.2.2 Emergency Stop Observation	△	△	△	△	○	□	□	479
15.5.2.3 Drive Safety Function								480
15.5.2.3.1 SLS (Safely-Limited Speed)	△	△	△	△	○	□	□	480
15.5.2.3.2 SLP (Safely-Limited Position)	△	△	△	△	○	□	□	480
15.5.2.3.3 SOS (Safe Operating Stop)	△	△	△	△	○	□	□	480
15.5.2.3.4 SSM (Safe Speed Monitor)	△	△	△	△	○	□	□	480
15.5.2.3.5 SBC/SBT (Safe Brake Control/Safe Brake Test)	△	△	△	△	○	□	□	481
15.5.2.3.6 SCA (Safe Cam)	△	△	△	△	○	□	□	481
15.5.2.3.7 SS1/SS2 (Safe Stop)	△	△	△	△	○	□	□	481
15.5.2.3.8 STO (Safe Torque Off)	△	△	△	△	○	□	□	481
16 Drive System								**483**
16.1 Servo/Spindle								**484**
16.1.1 Feed Axis								484
16.1.1.1 MDS-E-Vx	□	□	□	□	□	□	□	484
16.1.1.2 MDS-EH-Vx	□	□	□	□	□	□	□	484
16.1.1.3 MDS-EJ-Vx	□	□	□	□	□	□	□	484
16.1.1.4 MDS-EJH-Vx	□	□	□	□	□	□	□	484
16.1.1.5 MDS-EM-SPVx	□	□	□	□	□	□	□	484
16.1.1.6 MDS-EMH-SPVx	□	□	□	□	□	□	□	484

○ : Standard △ : Option □ : Selection

Class	Machining center system							Page
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB	
16.1.2 Spindle								485
16.1.2.1 MDS-E-SPx	□	□	□	□	□	□	□	485
16.1.2.2 MDS-EH-SPx	□	□	□	□	□	□	□	485
16.1.2.3 MDS-EJ-SPx	□	□	□	□	□	□	□	485
16.1.2.5 MDS-EM-SPVx	□	□	□	□	□	□	□	485
16.1.2.6 MDS-EMH-SPVx	□	□	□	□	□	□	□	485
16.1.4 Power Supply								485
16.1.4.1 MDS-E-CV	□	□	□	□	□	□	□	485
16.1.4.2 MDS-EH-CV	□	□	□	□	□	□	□	485
16.1.4.3 MDS-EX-CVP	□	□	□	□	□	□	□	485
17 Machine Support Functions								**487**
17.1 PLC								**488**
17.1.1 Built-in PLC Processing Mode	○	○	○	○	○	○	○	488
17.1.2 PLC Functions								488
17.1.2.1 Built-in PLC Basic Function	○	○	○	○	○	○	○	488
17.1.2.1.1 Index Modification	○	○	○	○	○	○	○	492
17.1.2.1.2 Multi-Program [Number of Programs]	○ 120	○ 120	○ 120	○ 120	○ 60	○ 60	○ 60	492
17.1.2.1.3 Multi-Project [Number of Projects Stored]								492
Number of PLC projects: 2	○	○	○	○	—	—	○	492
Number of PLC projects: 3	△	△	△	△	○	○	—	492
Number of PLC projects: 6	△	△	△	△	—	—	—	492
17.1.2.1.4 Function Block (FB)	○	○	○	○	○	○	○	492
17.1.2.1.5 Label Programming	○	○	○	○	○	○	○	492
17.1.2.1.6 ST Language	○	○	○	○	○	○	○	492
17.1.2.2 PLC Exclusive Instruction	○	○	○	○	○	○	○	493
17.1.2.3 Enhanced PLC Security Mode	○	○	○	○	○	○	○	494
17.1.3 PLC Support Functions								495
17.1.3.1 Alarm Message Display	○	○	○	○	○	○	○	495
17.1.3.2 Operator Message Display	○	○	○	○	○	○	○	495
17.1.3.3 Memory Switch (PLC Switch)								496
17.1.3.3.1 Memory Switch (PLC Switch) 32 Points	○	○	○	○	○	○	○	496
17.1.3.3.2 Memory Switch (PLC Switch) 64 Points	△	△	△	△	○	○	○	496
17.1.3.3.3 Memory Switch (PLC Switch) 96 Points	△	△	△	△	—	—	—	496
17.1.3.4 Load Meter Display	○	○	○	○	○	○	○	497
17.1.3.5 User PLC Version Display	○	○	○	○	○	○	○	497
17.1.3.6 Ladder Program Writing During RUN	○	○	○	○	○	○	○	497
17.1.3.7 PLC Program Protection	○	○	○	○	○	○	○	497
17.1.4 Built-in PLC Capacity								498
17.1.4.1 Standard PLC Capacity [Number of Steps]	○ 128K	○ 128K	○ 128K	○ 128K	○ 64K	○ 64K	○ 32K	498
17.1.4.2 Large PLC Capacity: 256 Ksteps	△	△	△	△	—	—	—	498
17.1.4.3 Large PLC Capacity: 512 Ksteps	△	△	△	△	—	—	—	498
17.1.5 Machine Contact Input/Output I/F	○	○	○	○	○	○	○	498
17.1.6 Ladder Monitor	○	○	○	○	○	○	○	499
17.1.7 PLC Development								500
17.1.7.1 On-Board Development	○	○	○	○	○	○	○	500
17.1.7.2 MELSEC Development Tool (GX Developer)	○	○	○	○	○	○	○	500
17.1.7.3 MELSEC Development Tool (GX Works2)	○	○	○	○	○	○	○	500
17.1.8 PLC Parameter								501
17.1.8.1 PLC Constant (150 Points)	○	○	○	○	○	○	○	501
17.1.8.2 PLC Constant Extension (Up to 750 Points)	○	○	○	○	○	○	○	501
17.1.8.3 Second PLC Constant Extension (300 Points)	○	○	○	○	○	○	○	501
17.1.9 GOT Connection								502

○ : Standard △ : Option □ : Selection

Class	Machining center system							Page
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB	
17.1.9.1 Ethernet Connection	○	○	○	○	○	○	○	502
17.1.9.2 CC-Link Connection	○	○	○	○	○	○	○	502
17.1.10 Pallet Program Registration	△	△	△	△	—	—	—	503
17.2 Machine Construction								**504**
17.2.1 Servo OFF	○	○	○	○	○	○	○	504
17.2.2 Axis Detachment	○	○	○	○	○	○	○	505
17.2.3 Synchronous Control	△	△	△	△	○	○	○	506
17.2.4 Inclined Axis Control	△	△	△	△	○	○	○	509
17.2.5 Position Switch	○ 24	○ 24	○ 24	○ 24	○ 24	○ 24	○ 24	510
17.2.7 Index Table Indexing	○	○	○	○	○	○	○	511
17.2.8 Tool Length Compensation Along the Tool Axis	△	△	△	△	—	—	—	513
17.2.9 Tool Handle Feed & Interruption	△	△	△	△	—	—	—	514
17.2.10 Tool Center Coordinate Display	△	△	△	△	—	—	—	514
17.2.11 Tool Center Point Control								515
17.2.11.1 Tool Center Point Control (G43.4/G43.5)	△	△ (*3)	△	△ (*3)	○ (*3)	○ (*3)	—	515
17.2.11.2 Tool Cutting Point Control Command (G43.8/G43.9)	△	—	△	—	—	—	—	517
17.2.12 Inclined Surface Machining Command	△	△	△	△	○	○	—	518
17.2.13 Simple Inclined Surface Machining Command	—	—	—	—	—	—	—	519
17.2.14 3-Dimensional Tool Radius Compensation (Tool's Vertical-Direction Compensation)	△	△	△	△	—	—	—	521
17.2.15 Workpiece Installation Error Compensation	△	—	△	—	—	—	—	522
17.2.16 3-Dimensional Manual Feed	△	△	△	△	○	○	—	523
17.2.17 R-Navi	△	△	△	△	○	○	—	524
17.2.18 Simple Synchronous Control	△	△	△	△	○	○	—	526
17.2.20 Real-Time Tuning								527
17.2.20.1 Real-Time Tuning 1 (Speed Gain)	△	△	△	△	○	○	—	527
17.2.20.2 Real-Time Tuning 2 (Rapid Traverse Time Constant)	△	△	△	△	○	○	—	528
17.2.21 Constant Torque Control	△	△	△	△	○	○	○	529
17.2.22 External Encoder Position Output I/F	○	○	○	○	○	○	○	530
17.2.23 Multiple-Axis Synchronization Control	△	△	△	△	○	○	○	531
17.2.25 BiSS Encoder I/F	—	—	○	○	—	○	○	532
17.2.26 Tool Head Hot Swapping	△	△	△	△	—	—	—	533
17.3 PLC Operation								**534**
17.3.1 Arbitrary Feed in Manual Mode	○	○	○	○	○	○	○	534
17.3.2 Circular Feed in Manual Mode	△	△	△	△	○	○	○	535
17.3.3 PLC Axis Control	○	○	○	○	○	○	○	537
17.3.5 PLC Axis Indexing	○	○	○	○	○	○	○	538
17.3.6 NC Axis/PLC Axis Switchover	△	△	△	△	○	○	○	540
17.4 PLC Interface								**541**
17.4.1 CNC Control Signal	○	○	○	○	○	○	○	541
17.4.2 CNC Status Signal	○	○	○	○	○	○	○	542
17.4.3 PLC Window	○	○	○	○	○	○	○	544
17.4.4 External Search	○	○	○	○	○	○	○	545
17.4.5 Direct Screen Selection	○	○	○	○	○	○	○	545
17.4.6 Buzzer Sound Control	○	○	○	○	○	○	○	546
17.4.7 Extended ZR Device	○	○	○	○	○	○	○	546
17.5 Machine Contact I/O								**547**
17.5.1 Operation Panel I/O								547
17.5.1.1 DI:64/DO:64	□	□	—	—	□	—	—	547
17.5.1.2 DI:64/DO:64 + SDI:8	□	□	—	—	□	—	—	547
17.5.1.3 DI:96/DO:64	—	—	□	□	—	□	□	547

○ : Standard △ : Option □ : Selection

Class	Machining center system							Page
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB	
17.5.1.5 DI:64/DO:48/AO:1	—	—	□	□	—	□	□	547
17.5.2 Remote I/O								548
17.5.2.1 DI:32/DO:32	□	□	□	□	□	□	□	548
17.5.2.2 DI:64/DO:48	□	□	□	□	□	□	□	548
17.5.2.3 DI:64/DO:48+AO:1	□	□	□	□	□	□	□	548
17.5.2.5 DI:16/DO:8	□	□	□	□	□	□	□	548
17.5.2.6 DI:32/DO:32 + SDI:8/SDO:4	□	□	□	□	□	□	□	548
17.5.2.7 SDI:8/SDO:4	□	□	□	□	□	□	□	548
17.6 External PLC Link								**549**
17.6.1 CC-Link (Master/Local)	□	□	□	□	□	□	□	549
17.6.2 PROFIBUS-DP (Master)	□	□	□	□	□	□	□	555
17.6.3 CC-Link IE Field (Master/Local)	□	□	□	□	□	□	□	556
17.6.5 EtherNet/IP	□	□	□	□	□	□	□	558
17.6.8 FL-Net	□	□	□	□	□	□	□	559
17.6.10 CC-Link IE Field Basic	○	○	○	○	○	○	○	560
17.6.11 CC-Link IE TSN (Remote Station)	□	□	□	□	□	□	□	562
17.7 Installing S/W for Machine Tools								**563**
17.7.1 Customization (NC Designer2)	△	△	△	△	○	○	○	563
17.7.1.1 Customization Data Storage Capacity [MB]	□	□	16	16	□	16	16	565
17.7.1.2 Customization Working Memory Size [MB]	6	6	6	6	6	6	6	565
17.7.2 User-Defined Key	○	○	○	○	○	○	○	565
17.7.3 EZSocket I/F	○	○	○	○	○	○	○	565
17.7.4 APLC Release	△	△	△	△	○	○	○	566
17.7.5 Custom API Library	○	○	○	○	○	○	○	567
17.7.6 MES Interface Library	△	△	△	△	○	○	○	567
17.7.7 SLMP Server	○	○	○	○	○	○	○	568
17.7.8 Numerical Control (CNC) Communication Software FCSB1224W000	○	○	○	○	○	○	○	569
17.7.11 Motion Control Release								570
17.7.11.1 Coordinate Transformation	△	—	△	—	—	—	—	570
17.7.12 Connection with High Speed Processing Unit (HPU)	□	□	—	—	□	—	—	571
17.7.13 Direct Socket Communication I/F	△	△	—	—	○	—	—	572
17.8 Others								**573**
17.8.1 System Lock	△	△	△	△	○	○	○	573
17.8.2 CNC Remote Operation Tool								573
17.8.2.1 NC Monitor2	○	○	○	○	○	○	○	573
17.8.2.2 NC Explorer	○	○	○	○	○	○	○	573
17.8.3 Automatic Operation Lock	○	○	○	○	○	○	○	574
17.8.4 Power Consumption Computation	○	○	○	○	○	○	○	574
17.8.5 EcoMonitorLight Connection	○	○	○	○	○	○	○	574
17.8.6 GOT Window	○	○	○	○	○	○	○	575
17.8.8 Laser Processing Control	□	□	□	□	□	□	□	576
17.8.9 Wireless LAN	—	—	○	○	—	○	○	578
18 Appendix 1: Functions with Advanced Machining								**579**

(*2) The 3rd part system is the loader-dedicated part system.

(*3) Restrained to 4-axis simultaneous contouring control.

1

Control Axes

1.1 Control Axes

The NC axis, spindle, PLC axis and auxiliary axis are generically called the control axis.

NC axis	The axis is used to perform automatic operation by coordinate value command from machining program and to perform manual operation such as jog feed, handle feed etc. X, Y, Z, U, V, W, A, B and C can be used as the axis name. An axis where no motor is connected (hypothetical axis) can be set.
Spindle	The axis is used to rotate the part where the tool/workpiece is attached. The rotation is commanded by rotation speed and spindle forward run signal/spindle reverse run signal.
PLC axis	The axis is used to command the position and speed from user PLC. Two or more axes are not used together for interpolation; instead, each axis is used individually. Manual operation such as jog feed, handle feed, etc. can be performed. An axis where no motor is connected (hypothetical axis) can be set.

The following functions can be added depending on the parameter setting.

- Spindle position control (spindle/C axis control)
  A spindle can be controlled as the rotary axis. A coordinate value (angle) can be commanded from machining program in the same way as NC axis.
- Spindle-mode rotary axis control
  NC axis can be controlled by the command of rotation speed and spindle forward run signal/spindle reverse run signal in the same way as spindle.

Note

(1) The number of control axes that can be controlled by one CNC depends on the model.

1.1.1 Number of Basic Control Axes (NC Axes)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○ 3	○ 3	○ 3	○ 3	○ 3	○ 3	○ 3
L	○ 2	○ 2	○ 2	○ 2	○ 2	○ 2	○ 2

1.1.2 Max. Number of Axes (NC Axes + Spindles + PLC Axes)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○ 16 Δ32	○ 16 Δ32	○ 16 Δ32	○ 16 Δ32	24	24	12
L	○ 16 Δ32	○ 16 Δ32	○ 16 Δ32	○ 16 Δ32	24	24	9

A number of axes that are within the maximum number of axes, and that does not exceed the maximum number given for the NC axis, spindle and PLC axis can be used.

■ Connection specifications of NC axis, PLC axis and spindle:

NC axes, PLC axes and spindles (except for analog spindles) are connected to the high-speed optical servo communication (OPTH).
Refer to the "Connection and Setup Manual" for details.

1.1.2.1 Max. Number of NC Axes (In Total for All the Part Systems)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○ 16 Δ32	○ 16 Δ32	○ 16 Δ32	○ 16 Δ32	16	16	6
L	○ 16 Δ32	○ 16 Δ32	○ 16 Δ32	○ 16 Δ32	16	16	7

1.1.2.2 Max. Number of Spindles

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	6	6	6	6	4	4	3
L	8	8	8	8	6	6	4

Includes analog spindles.

1.1.2.3 Max. Number of PLC Axes

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	8	8	8	8	6	6	6
L	8	8	8	8	6	6	6

1.1.4 Max. Number of Indexing PLC Axes

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	8	8	8	8	4	4	4
L	8	8	8	8	4	4	4

PLC axis indexing is used to move the PLC axis to the positioning destination or an arbitrary coordinate position.

1.1.5 Number of Simultaneous Contouring Control Axes

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	8	4	8	4	4	4	4
L	8	4	8	4	4	4	4

Simultaneous control of all axes is possible as a principle in the same part system. However, for actual use, the machine tool builder specification will apply.

1.1.6 Max. Number of NC Axes in a Part System

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○ 8 Δ12 Δ16	○ 8 Δ12 Δ16	○ 8 Δ12 Δ16	○ 8 Δ12 Δ16	8	8	6
L	○ 8 Δ12 Δ16	○ 8 Δ12 Δ16	○ 8 Δ12 Δ16	○ 8 Δ12 Δ16	8	8	5

Listed are the maximum number of axes which can be controlled in a part system. However, for actual use, the machine tool builder specification will apply.

1.1.7 Axis Name Extension

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

The axis name (command axis name) to perform the absolute/incremental command to NC control axis can be expanded to two letters. Because the command axis name while this function is invalid is set with one letter from A, B, C, H (*1), U, V, W, X, Y or Z, when the incremental command axis name is used (two letters of the alphabet are used per axis), the number of axes is limited. The incremental command axis name can be used to all axes by this function. The name extension axis cannot be designated to the parameter which sets the command axis name such as axes configuring plane I, J or K. Thus, apply this function to miscellaneous axis which is not used for machining (cutting).

(*1) "H" can be set only for the L system.

1.2 Control Part System

1.2.1 Standard Number of Part Systems

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	1	1	1	1	1	1	1
L	1	1	1	1	1	1	1

1.2.2 Max. Number of Part Systems (Main + Sub)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○ 2 Δ 3 (*1) Δ4 Δ8	○ 2 Δ 3 (*1) Δ4 Δ8	○ 2 Δ 3 (*1) Δ4 Δ8	○ 2 Δ 3 (*1) Δ4 Δ8	○ 4	○ 4	○ 2
L	○ 4 Δ8	○ 4 Δ8	○ 4 Δ8	○ 4 Δ8	○ 4	○ 4	○ 2

(*1) The 3rd part system is the loader-dedicated part system.

For actual use, the machine tool builder specification will apply.

1.2.2.1 Max. Number of Main Part Systems

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○ 2 Δ4 Δ8	○ 2 Δ4 Δ8	○ 2 Δ4 Δ8	○ 2 Δ4 Δ8	○ 4	○ 4	○ 1
L	○ 4 Δ8	○ 4 Δ8	○ 4 Δ8	○ 4 Δ8	○ 3	○ 3	○ 2

1.2.2.2 Max. Number of Sub Part Systems

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○ 2 Δ4 Δ8	○ 2 Δ4 Δ8	○ 2 Δ4 Δ8	○ 2 Δ4 Δ8	○ 2	○ 2	○ 1
L	○ 4 Δ8	○ 4 Δ8	○ 4 Δ8	○ 4 Δ8	○ 2	○ 2	○ 1

1.3 Control Axes and Operation Modes

1.3.1 Tape (RS-232C Input) Mode

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

In this mode, operation is performed using the machining program data from the RS-232C interface built in the CNC unit. A paper tape reader must be provided if machining programs on paper tape are to be run.

1.3.2 Memory Mode

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

The machining programs stored in the memory of the CNC unit are run.

1.3.3 MDI Mode

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

The MDI data stored in the memory of the CNC unit is executed. Once executed, the MDI data is set to the "setting incomplete" status, and the data will not be executed unless the "setting completed" status is established by screen operations.

1.3.4 High-Speed Program Server Mode

1.3.4.1 Control Unit-Side High-Speed Program Server Mode

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	—	—	○	—	—
L	Δ	Δ	—	—	○	—	—

The machining program stored in SD card can be operated by installing a SD card in the control unit SD card interface. From this, mass capacity and multi-process machining can be done.
Machining programs can be copied to SD card with the front SD card or Ethernet on the input/output screen.

When a machining program stored in SD card is searched while "DS" (Data server) is selected for device during operation search, the machining program in SD card can be operated as a main program. ("Memory mode" is used as the operation mode.)

The machining program in SD card can be called and operated with the M198 command.

Macros such as WHILE, IF and GOTO can be used. Also, calling the sub program and macro program stored in memory or SD card is possible.

1.3.4.2 Display Unit-Side High-Speed Program Server Mode

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

The machining program stored in the built-in disk of the display unit can be operated.
The built-in disk of the display unit is mounted in the personal computer for M800VW/M80VW.
For M800VS/M80V, the SD card inserted into SD card I/F on the back of the display unit is equivalent to the built-in disk of the display unit.

When a machining program stored in the built-in disk of the display unit is searched while "HD" (M800VW/M80VW) or "DS" (M800VS/M80V) is selected for device during operation search, the machining program in the built-in disk of the display unit can be operated as a main program. ("Memory mode" is used as the operation mode.)

The machining program in the built-in disk of the display unit can be called and operated with the M98 or G65/G66/G66.1 command depending on the parameter setting. (*1)

Macros such as WHILE, IF and GOTO can be used. Also, calling the sub program and macro program stored in memory or the built-in disk of the display unit is possible.

(*1) The G65/G66/G66.1 commands are available for M800VW/M80VW Series only.

1.3.5 Front-Side SD Card Mode

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

The machining program stored in a SD card can be operated. This SD card is installed to the front-side SD card I/F.

When a machining program stored in SD card is searched while "Memory Card" is selected for device during operation search, the machining program in SD card can be operated as a main program. ("Memory mode" is used as the operation mode.)

The machining program in SD card can be called and operated with the M98 command depending on the parameter setting.

Macros such as WHILE, IF and GOTO can be used. Also, calling the sub program and macro program stored in memory or SD card is possible.

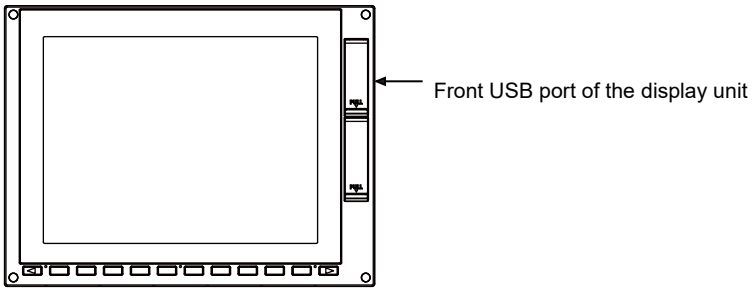
1.3.6 Front-Side USB Memory Mode

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

The control is able to run a machining program stored in a USB memory when it is inserted into the front USB port of the display unit.
USB memory may be pulled off due to the machine vibration depending on the environment. Use this function at your own risk.
USB memory mode enables you to handle the same programs as for memory mode. In addition you can handle the greater number of large-capacity programs.
When a machining program stored in the USB memory is searched while USB is selected for device during operation search, the machining program in the USB memory can be operated as a main program.
("Memory mode" is used as the operation mode.)

The machining program in the USB memory can be called and operated with the M98 command depending on the parameter setting.

[Display unit]



2

Input Command

2.1 Data Increment

2.1.1 Least Command Increment

[M system]

Least command increment (input setting increment)	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
1 μm (1 μm)	○	○	○	○	○	○	○
0.1 μm (0.1 μm)	○	○	○	○	○	○	○
0.01 μm (10 nm)	Δ	Δ	Δ	Δ	—	—	—
0.001 μm (1 nm)	Δ	Δ	Δ	Δ	—	—	—

[L system]

Least command increment (input setting increment)	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
1 μm (1 μm)	○	○	○	○	○	○	○
0.1 μm (0.1 μm)	○	○	○	○	○	○	○
0.01 μm (10 nm)	Δ	Δ	Δ	Δ	—	—	—
0.001 μm (1 nm)	Δ	Δ	Δ	Δ	—	—	—

The data increment handled in the controller includes the input setting increment, PLC increment and machine error compensation increment. Each type is set with parameters.

Input setting increment

The input setting increment applies to the data entered from or displayed in the Monitor or other screens, the travel distance per pulse of manual pulse generator, machining programs, and maintenance data. (Excluding PLC increment and machine error compensation increment mentioned below.) This increment is applied per part system (1st to the maximum part system within the specification, PLC axis).

Input setting increment (parameter)	Metric unit system		Inch unit system	
	Linear axis (Unit = mm)	Rotary axis (Unit = °)	Linear axis (Unit = inch)	Rotary axis (Unit = °)
1 μm (B)	0.001	0.001	0.0001	0.001
0.1 μm (C)	0.0001	0.0001	0.00001	0.0001
10 nm (D)	0.00001	0.00001	0.000001	0.00001
1 nm (E)	0.000001	0.000001	0.0000001	0.000001

Note

(1) The inch and metric systems cannot be used together.

PLC increment

PLC increment is used for setting and display of PLC interface. Input-output to ladder is handled with this increment. (excluding machine error compensation increment) This increment is applied per part system.

(Example) When writing "90°" to A axis and "180°" to C axis in the A-C axis configuration using the rotary axis angle designation(*1)

[When PLC increment is 1 μm (B)]

	Angle		1/1000°		System unit	
<A axis>	90	×	1000	=	90000	┤├[DMOV K90000 R2628]─
<C axis>	180	×	1000	=	180000	┤├[DMOV K180000 R2630]─

[When PLC increment is 1 nm (E)]

	Angle		1/1000000°		System unit	
<A axis>	90	×	1000000	=	90000000	┤├[DMOV K90000000 R2628]─
<C axis>	180	×	1000000	=	180000000	┤├[DMOV K180000000 R2630]─

(*1) Set the angle of the rotary axis of the mechanical axis specification.
(When the PLC unit is 1 μm (B))

R register	Details	Input range
R2628, R2629	The angle of the 1st rotary axis with mechanical axis specifications (1st part system)	0 to ± 720000 (1°/1000)
R2630, R2631	The angle of the 2nd rotary axis with mechanical axis specifications (1st part system)	

Machine error compensation increment

Machine error compensation increment is used for setting and display of machine error compensation amount. This increment is applied per part system.
This increment is applied for following parameters and PLC interface.

- Backlash compensation (parameter)
  This function compensates for the error (backlash) produced when the direction is reversed.
- Pitch error compensation (parameter)
  This function compensates for the errors in pitch intervals of ball screws.
- Machine rotation center error compensation (parameter)
  This function compensates for the errors difference between the center of the actual rotary axis and the center of programmed rotary axis.
- External machine coordinate system compensation (PLC interface)
  This function shifts the coordinate system by PLC.
- Ball screw thermal displacement compensation (PLC interface)
  This function compensates for axis feed error caused by the ball screw thermal expansion, etc.

2.1.2 Least Control Increment

[M system]

Least control increment	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
0.01 μm (10 nm)	O	O	O	O	O	O	O
0.001 μm (1 nm)	O	O	O	O	O	O	O

[L system]

Least control increment	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
0.01 μm (10 nm)	O	O	O	O	O	O	O
0.001 μm (1 nm)	O	O	O	O	O	O	O

The least control increment determines the CNC's internal operation accuracy.
The control increment (the unit of the NC internal position data) and output increment (the unit of the data communicated to the drive unit) can be set individually.

2.1.3 Indexing Increment

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

This function limits the command value for the rotary axis.
This can be used for indexing the rotary table, etc. It is possible to cause a program error with a program command other than an indexing increment (parameter setting value).

(Example) When the indexing increment setting value is 2 degrees, only command with the 2-degree increment are possible.

G90 G01 C102.000 ;	Moves to the 102 degree angle.
G90 G01 C101.000 ;	Program error
G90 G01 C102 ;	Moves to the 102 degree angle. (Decimal point type II)

2.2 Unit System

2.2.1 Inch/Metric Changeover

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

The unit systems of the data handled in the controller include the metric unit system and inch unit system. The unit (inch/mm) for the setting and display, as well as for the handle/incremental feed can be switched with either the parameters or machining program (G20/G21 command).
An optional specification is required when the unit is switched with the machining program command.
The command increment of linear axis is inch system during G20 modal, and metric system during G21 modal.
The command increment of rotary axis is "degree" regardless of G20 or G21 modal.

Example of data by unit system

Unit system	Data	Meaning
Metric unit system	1.0	1.0 mm or 1°(degree)
Inch unit system	1.0	1.0 inch or 1°(degree)

The unit system can be switched as follows with either the parameters or machining program.

Parameters		Data			
		Machining program		Screen data (Compensation amount, user parameter, counter, etc.) / Feedrate of handle, etc.	Machine parameter/PLC interface machine position, etc.
#1041 I_inch Select initial inch/mm	0	G20	Inch unit system	Metric unit system	Not affected
		G21	Metric unit system		
	1	G20	Inch unit system	Inch unit system	
		G21	Metric unit system		
#1040 M_inch Constant input (inch)	0	Not affected		Not affected	Metric unit system
	1				Inch unit system

Note

(1) The parameter changeover is valid after the power is turned ON again.
(2) The unit system for the PLC axis can be switched with a parameter different from the one used with the NC axis. The PLC axis unit system cannot be switched with the machining program (G20/G21 command).
(3) When the power is turned ON or resetting is performed, the command increment depends on the parameter setting.

2.2.2 Input Command Increment Tenfold

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	—	—	—	—	—	—	—

The program's command increment can be multiplied by an arbitrary scale with the parameter designation.
This function is valid when a decimal point is not used for the command increment.

For example, this function allows a CNC unit, for which the command increment is set to 1 μm, to run a machining program, which has been created with a 10 μm input command increment, as same as before.
The scale is set with the parameters.

Note

(1) This function cannot be used for the dwell function G04_X_(P_);.
(2) This function cannot be used for the compensation amount of the tool offset input.
(3) This function can be used when decimal point type I is valid, but cannot be used when decimal point type II is valid.
(4) This function cannot be used for a tool shape setting command (in G10L100 format).

2.3 Program Format

2.3.1 Program Format

This is G code (program) format.
The G-code of lathe system is selected by parameter.
This manual explains the G function with G-code list 3 as standard.

2.3.1.1 Format 1 for Lathe (G Code List 2, 3)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	○	○	○	○	○	○	○

2.3.1.2 Format 2 for Lathe (G Code List 4, 5)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	○	○	○	○	○	○	○

2.3.1.3 Special Format for Lathe (G Code List 6, 7)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	○	○	○	○	○	○	○

2.3.1.4 Format 1 for Machining Center

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	—	—	—	—	—	—	—

2.3.1.5 Format 2 for Machining Center (M2 Format)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	—	—	—	—	—	—	—

2.3.1.6 MITSUBISHI CNC Special Format

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	○	○	○	○	○	○	○

The formats of the turning fixed cycles (G77 to G79), compound type fixed cycle for turning machining (G71 to G76) and the fixed cycle for drilling (G80 to G89) can be switched to the MITSUBISHI CNC special formats.

Note

(1) There is a specification (zigzag thread cutting) for specific machine tool builder.

2.3.2 Program Format Switch

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	Δ	Δ	Δ	Δ	—	—	—

Program format switch is a function designed to switch the program format (G code list) using G codes or PLC signal. While the switch is active, the control runs a program based on the switched G code list. When you run a lathe-based multi-tasking machine, and if you change to the G code list of Machining center system, you can use a free-curved surface machining program made with CAM without modifying the program.

[Switchover using G codes (G code method)]

■ Command format

Program format switch ON

When G188 is given, the control switches the L-system program format to M-system (G code list 1 (Command type II)). The control runs the next and subsequent program blocks based on the M-system G code list.

G188 ; Change to M-system G code list (G code list 1 (Command type II))

Program format switch cancel

When G189 is given, the control switches the M-system program format back to L-system (a system selected by command type parameter). The control runs the next and subsequent program blocks based on the L-system G code list.

G189 ; Change back to L-system G code list (G code list determined by command type parameter)

■ Program example

<For G code list 2>

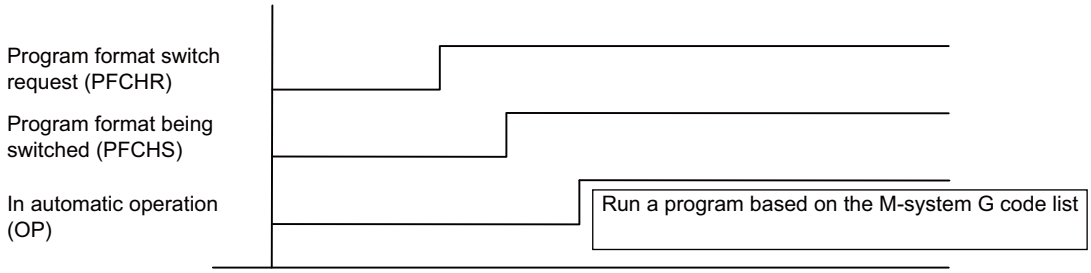
O1 : 	L system (G code list 2 (Command type III)
G188;	Program format switch ON
G90 G00 X20. Z200.; G91 G01 X30. Z180. F100; :	M system (G code list 1 (Command type II))
G189;	Program format switch cancel
: 	L system (G code list 2 (Command type III)

Switchover using PLC signal (PLC I/F method)

PFCHR ON;	**Change to M-system G code list (G code list 1 (Command type II))**
PFCHR OFF;	**Change back to L-system G code list (G code list determined by command type parameter)**

Note

(1) A switchover using the PLC I/F method is unavailable during automatic operation.



G code method and PLC I/F method are distinguished in the following points:

G code method	This switches by G188/G189 commands in cycle operation. (e.g. to switch the format dynamically for some machining processes)
PLC I/F method	This switches by PLC signal not in cycle operation. (e.g. to enable the switched state at the time of power up)

2.4 Command Value

2.4.1 Decimal Point Input I, II

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

There are two types of the decimal point input commands and they can be selected by parameter.

■ Decimal point input type I

When axis coordinates and other data are issued in machining program commands, the assignment of the program data can be simplified by using a decimal point. The minimum digit of a command not using a decimal point is the same as the least command increment.
The decimal point can be applied not only to axis coordinate position but also to speed commands and dwell commands. The decimal point position serves as the millimeter unit in the metric mode, as the inch unit in the inch mode and as the second unit in a time designation of dwell command.

■ Decimal point input type II

As opposed to type I, the minimum digit of a command without a decimal point serves as the millimeter unit in the metric mode, as the inch unit in the inch mode and as the second unit in the time designation.
The "." (point) must be added when commands below the decimal point are required.

	Unit interpretation (for metric system)	
	Type I	Type II
G00 X100. Y-200.5	X: 100 mm Y: -200.5 mm	X: 100 mm Y: -200.5 mm
G1 X100 F20.	X100 μm, F20 mm/min	X100 mm, F20 mm/min
G1 Y200 F100 (Note 1)	Y200 μm, F100 mm/min	Y200 mm, F100 mm/min
G4 X1.5	Dwell 1.5 s	Dwell 1.5 s
G4 X2	Dwell 2 ms	Dwell 2 s

Note

(1) The F unit is mm/min for either type (inch system : inch/min).

2.4.2 Absolute/Incremental Command

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

[M system]

When axis coordinate data is issued in a machining program command, either the incremental command method (G91) that commands a relative distance from the current position or the absolute command method (G90) that moves to a designated position in a predetermined coordinate system can be selected.

The absolute and incremental commands can be both used in one block, and are switched with G90 or G91. However, the arc radius designation (R) and arc center designation (I, J, K) always use incremental designations.

The absolute and incremental commands are switched with G190 and G191 depending on G code list.

G90, G190	Absolute command
G91, G191	Incremental command

These G codes can be commanded multiple times in one block.

[L system]

When axis coordinate data is issued in a machining program command, either the incremental command method that commands a relative distance from the current position or the absolute command method that moves to a designated position in a predetermined coordinate system can be selected.

When issuing an incremental command, register the axis address to be commanded as the incremental axis name in the parameter. However, the arc radius designation (R) and arc center designation (I, J, K) always use incremental designations.

Absolute command	X, Z
Incremental command	U, W

Note

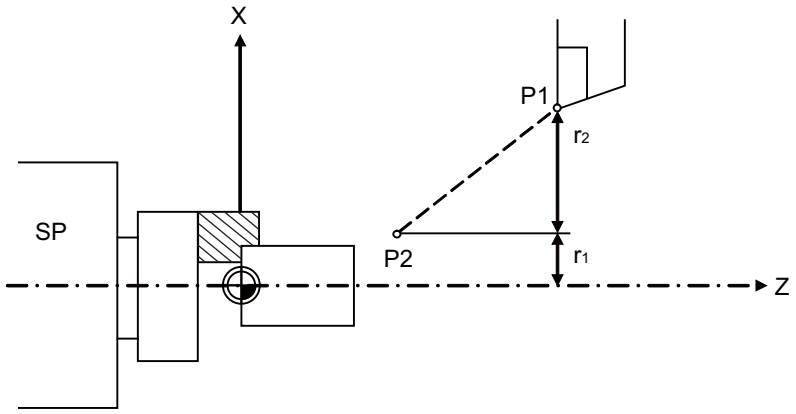
(1) Absolute command and incremental command can be switched by the parameter. In addition to the command method using the axis addresses as indicated above, a command method using G code (G90/G91 or G190/G191) may be selected.

2.4.3 Diameter/Radius Designation

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	O	O	O	O	O	O	O

For axis command value, the radius designation or diameter designation can be changed over with parameters.
When the diameter designation is selected, the scale of the length of the selected axis is doubled.
(Only the half (1/2) of the commanded amount moves.)
This function is used when programming the workpiece dimensions on a lathe as diameters.
Changing over from the diameter designation to the radius designation or vice versa can be set separately for each axis.

When the tool is to be moved from point P1 to point P2



SP Spindle

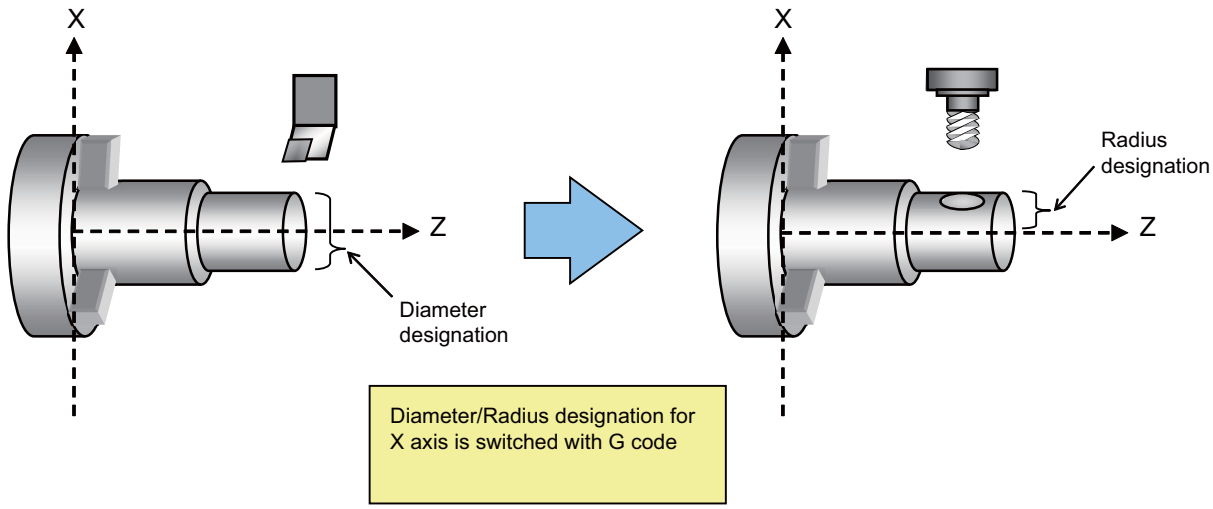
Radius and diameter commands

X command		U command		Remarks
Radius	Diameter	Radius	Diameter	Even when a diameter command has been selected, only the U command can be made a radius command by parameter.
$X = r_1$	$X = 2r_1$	$U = r_2$	$U = 2r_2$	

2.4.4 Diameter/Radius Designation Switch

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	—	—	—
L	O	O	O	O	O	O	O

The method of commanding a travel distance (command with a diameter dimension/command with a radius dimension (as-is distance)) in a program is defined individually for each axis using the parameter. Diameter/Radius designation switch function; however, enables you to switch the diameter/radius designation of each axis using a G code at your desired timing. When you use the function to switch diameter/radius designation, it helps you create a program more flexibly, from turning to milling, according to each machining situation.



■ Command format

Diameter/Radius Designation Switch

Diameter/Radius designation switch enables you to select any desired NC axis, excluding rotary axes, and switch the diameter/radius designation of the axis.

G10.9 Axis address 1__ Axis address 2__ ... Axis address n__ ;

Axis address n : Axis address for which diameter/radius designation is switched
Select radius or diameter designation with a value that follows the axis address.
0: Radius designation
1: Diameter designation

If you select no axis address, the diameter/radius designation statuses of all the axes of the part system are returned to the initial power-on state.

3

Positioning/Interpolation

3.1 Positioning

3.1.1 Positioning

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

This function carries out high-speed positioning following the movement command given in a program.

G00 Xx1 Yy1 Zz1 ;(Also possible for additional axes A, B, C, U, V, W simultaneously)
Xx1, Yy1, Zz1 :Position data

The above command positions the tool with rapid traverse rate. The tool path takes the shortest distance to the end point in the form of a straight line.
For details on the feedrate of the NC, refer to " 4.1.1 Rapid Traverse Rate (m/min)".
Since the actual rapid traverse feedrate depends on the machine, refer to the specifications of the machine concerned.

- The rapid traverse feed rate can be set for each axis with parameters.
- The number of axes which can be commanded simultaneously depends on the specifications (number of simultaneous contouring control axes). The axes can be used in any combination within this range.
- The feed rate is controlled within the range that it does not exceed the rapid traverse rate of each axis and so that the shortest time is taken. (Linear type)
  Parameter setting enables movement at the rapid traverse rates of the respective axes independently for each axis. In this case, the tool path does not take the form of a straight line to the end point. (Non-Linear type)
- The tool is always accelerated at the start of the program command block and decelerated at the end of the block.

3.1.2 Unidirectional Positioning

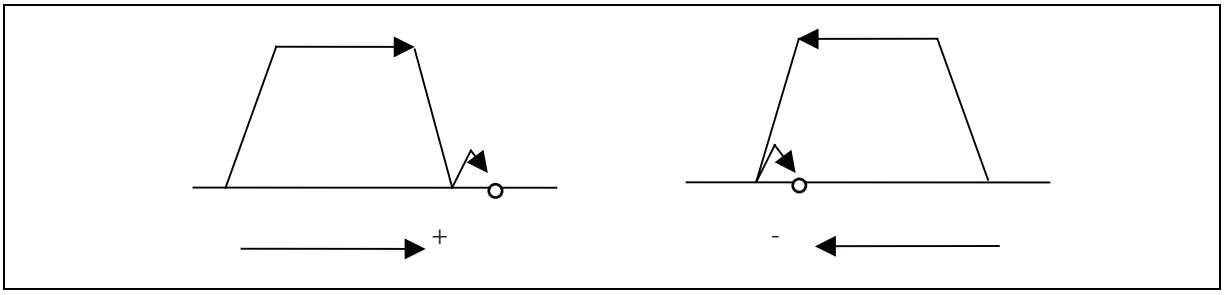
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	—	—	—	—	—	—	—

The G60 command always moves the tool to the final position in a direction determined with parameters.
The tool can be positioned without backlash.
The parameter setting enables G60 to switch between G code group 0 (non-modal) and G code group 01 (modal).

G60 X__ Y__ Z__ ;	**(Also possible for additional axes A/B/C/U/V/W simultaneously)**

X/Y/Z	Position data

With the above command, the tool is first moved to a position distanced from the end point by an amount equivalent to the creep distance (parameter setting) with rapid traverse and then moved to its final position.
For details on the feedrate of the NC, refer to " 4.1.1 Rapid Traverse Rate (m/min)".
Since the actual rapid traverse feed rate depends on the machine, refer to the specifications of the machine concerned.
Positioning to the final point is shown below (when this positioning is in the "+" direction.)



Note

(1) The processing of the above pattern will be followed even for the machine lock and Z-axis command cancel.
(2) On the creep distance, the tool is moved with rapid traverse.
(3) G60 is valid even for positioning in drilling in the fixed cycle.
(4) When the mirror image function is on, the tool will be moved in the reverse direction by mirror image as far as the interim position, but operation over the creep distance with the final advance will not be affected by the mirror image.

3.2 Linear/Circular Interpolation

3.2.1 Linear Interpolation

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

Linear interpolation is a function that moves a tool linearly by the movement command value supplied in the program at the cutting feed rate designated by the F code.

G01 Xx1 Yy1 Zz1 Ff1 ; (Also possible for additional axes A, B, C, U, V, W simultaneously)

Xx1,Yy1,z1	Position data
Ff1	Feedrate data

Linear interpolation is executed by the above command at the f1 feed rate. The tool path takes the shortest distance to the end point in the form of a straight line.
For details on the f1 command values for NC, refer to " 4.1.2 Cutting Feedrate (m/min)".
Since the actual cutting feed rate depends on the machine, refer to the specifications of the machine concerned.

- The number of axes which can be commanded simultaneously depends on the specifications (number of simultaneous contouring control axes). The axes can be used in any combination within this range.
- The feed rate is controlled so that it does not exceed the cutting feed rate clamp of each axis.
- When a rotary axis has been commanded in the same block, it is treated as a linear axis in degree(°) units (1° = 1mm), and linear interpolation is performed.

3.2.2 Circular Interpolation (Center/Radius Designation)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

Circular interpolation with I, J, K commands

This function moves a tool along a circular arc on the plane selected by the plane selection G code with movement command supplied in the program.

G02(G03) Xx1 Yy1 Ii1 Jj1 Ff1 ; **(Also possible for additional axes A, B, C, U, V, W)**

G02,G03	Arc rotation direction
Xx1,Yy1	End point coordinates
Ii1,Jj1	Arc center
Ff1	Feedrate

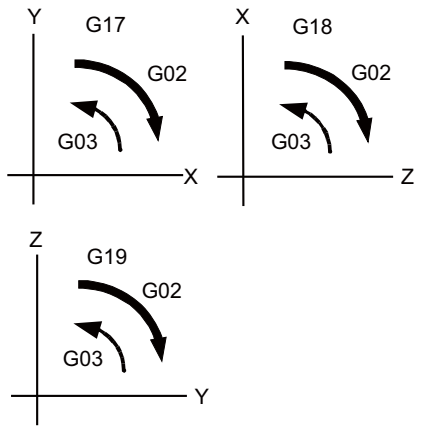
The above commands move the tool along the circular arc at the f1 feed rate. The tool moves along a circular path, whose center is the position from the start point designated by distance "i1" in the X-axis direction and distance "j1" in the Y-axis direction, toward the end point.

The direction of the arc rotation is designated by G02 or G03.

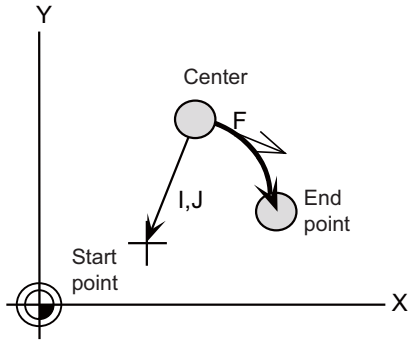
G02	Clockwise (CW)
G03	Counterclockwise (CCW)

The arc plane is designated by G17, G18 or G19.

G17	XY plane
G18	ZX plane
G19	YZ plane



(Example) See below for examples of circular commands.



- The axes that can be commanded simultaneously are the two axes for the selected plane.
- The feed rate is controlled so that the tool always moves at a speed along the circumference of the circle.
- Circular interpolation can be commanded within a range extending from 0° to 360°.
- A maximum of 6 digits before decimal point can be specified for the radius.

Note

(1) The arc plane is always based on the G17, G18 or G19 command. If a command is issued with two addresses which do not match the plane, an alarm will occur.
(2) The axes configuring a plane can be designated by parameters. Refer to " 10.1.9 Plane Selection".

R-specified circular interpolation

Besides the designation of the arc center coordinates using the above-mentioned I, J and K commands, arc commands can also be issued by designating the arc radius directly.

G02(G03) Xx1 Yy1 Rr1 Ff1 ; (Also possible for additional axes A, B, C, U, V, W)

G02,G03	Arc rotation direction
Xx1,Yy1	End point coordinates
Rr1	Arc radius
Ff1	Feedrate

The direction of the arc rotation is designated by G02 or G03.
The arc plane is designated by G17, G18 or G19.

The arc center is on the bisector which orthogonally intersects the segment connecting the start and end points, and the point of intersection with the circle, whose radius has been designated with the start point serving as the center, is the center coordinate of the arc command.
When the sign of the value of R in the command program is positive, the command will be for an arc of 180 or less; when it is negative, it will be for an arc exceeding 180.

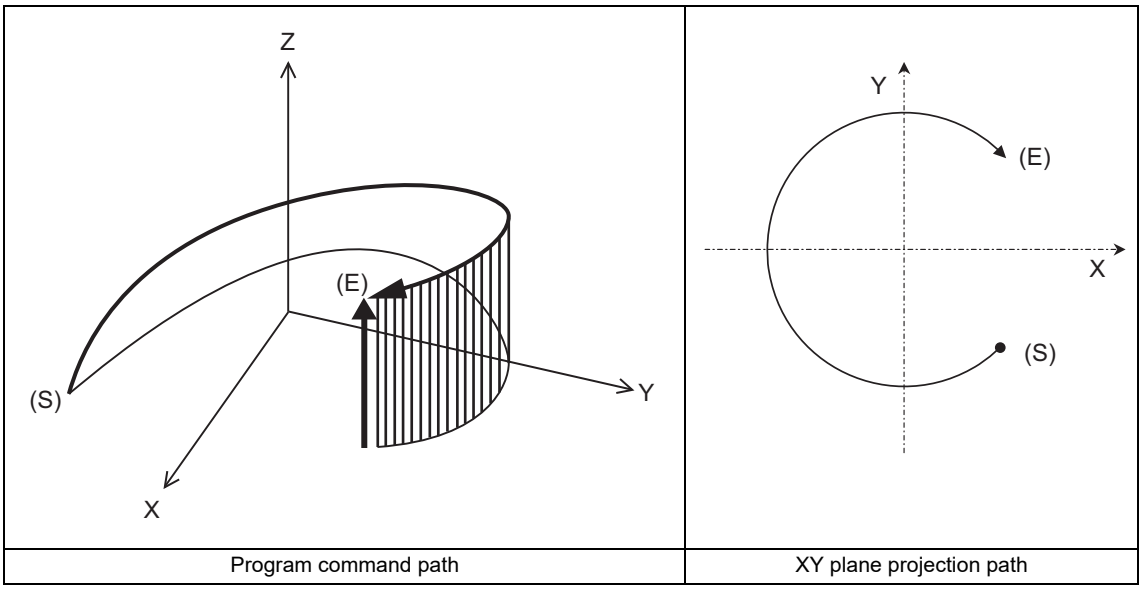
Note

(1) The arc plane is always based on the G17, G18 or G19 command. If a command is issued with two addresses which do not match the plane, an alarm will occur.

3.2.3 Helical Interpolation

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

With this function, any two of three axes intersecting orthogonally are made to perform circular interpolation while the third axis performs linear interpolation in synchronization with the arc rotation. This simultaneous 3-axis control can move helically to machine and be exercised to machine large-diameter screws or 3-dimensional cams.
This is commanded by combining a height axis to the circular interpolation command.



Program command path | XY plane projection path

(S) Start point
(E) End point

Program command path

Circular interpolation components

Linear interpolation components

■ Command format

[M system]

G17 G02/G03 X__ Y__ Z__ I__ J__ P__ F__ ;	**Helical interpolation command (Specify arc center)**

G17 G02/G03 X__ Y__ Z__ R__ F__ ;	**Helical interpolation command (Specify radius (R))**

G17/G18/G19	Arc plane (G17: XY plane, G18: ZX plane, G19: YZ plane)
G02/G03	Arc rotation direction (G02: clockwise, G03: counterclockwise)
X,Y	Arc end point coordinates
Z	Linear axis end point coordinates
I,J	Arc center coordinate values
R	Arc radius
P	Number of pitches
F	Feedrate

The arc center coordinate values and arc radius are commanded by the input increment. Pay attention to the case of the helical interpolation command of the axis which has a different input command increment.
To prevent confusion, command with a decimal point.
Absolute and incremental commands can be used for the arc end point coordinates and the end point coordinates of the linear axis, but incremental commands must be used for the arc center coordinates.
When executing the pitch command with ",P" address, the program error occurs.

[L system]

G17 G02/G03 X/U__ Y/V__ Z/W__ I__ J__ P/,P__ F__ ;

G17 G02/G03 X/U__ Y/V__ Z/W__ R__ F__ ;

G17/G18/G19	Arc plane (G17: XY plane, G18: ZX plane, G19: YZ plane)
G02/G03	Arc rotation direction (G02: clockwise, G03: counterclockwise)
X/U, Y/V	Arc end point coordinates
Z/W	Linear axis end point coordinates
I,J	Arc center coordinate values
R	Arc radius
P/,P	Number of pitches
F	Feedrate

Note

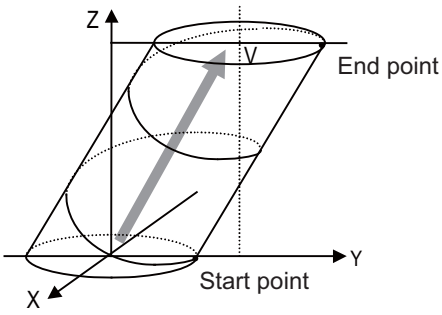
(1) Indicates as I axis: X, J axis: Y, and K axis: Z.
(2) When commanding "P" and ",P" for the number of pitches together, the command value ",P" is prioritized.

- The arc plane is designated by G17, G18 or G19.
- The direction of the arc rotation is designated by G02 or G03.
- Absolute and incremental commands can be used for the arc end point coordinates and the end point coordinates of the linear axis, but incremental commands must be used for the arc center coordinates.
- The linear interpolation axis is the other axis which is not included in the plane selection.
- Command the speed in the component direction that represents all the axes combined for the feed rate.
  The combination of the axes which can be commanded simultaneously depends on the specifications. The axes can be used in any combination under the specifications.
  The feed rate is controlled so that the tool always moves at a speed along the circumference of the circle.

Note

(1) Helical shapes are machined by assigning linear commands for one axis which is not a circular interpolation axis using an orthogonal coordinate system. It is also possible to assign these commands to two or more axes which are not circular interpolation axes.

When a simultaneous 4-axis command is used with the V axis as the axis parallel to the Y axis, helical interpolation will be carried out for a cylinder which is inclined as illustrated below. In other words, linear interpolation of the Z and V axes is carried out in synchronization with the circular interpolation on the XY plane.



3.2.4 Spiral/Conical Interpolation

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	—
L	—	—	—	—	—	—	—

This function interpolates arcs where the start point and end point are not on the circumference of the same circle into spiral shapes.
Conical interpolation is performed by commanding end point in height direction.
There are two types of command formats which can be changed with the parameters.

For command format type 1

■ Spiral interpolation

G02.1(G03.1) Xx1 Yy1 Ii1 Jj1 Pp1 Ff1 ;	
G02.1,G03.1	: Arc rotation direction
Xx1,Yy1	: End point coordinate
Ii1,Jj1	: Arc center
Pp1	: Number of pitches
Ff1	: Feed rate

The above commands move the tool along the circular arc at the f1 feed rate. The tool draws a spiral arc path whose center is at the position from the start point which is designated by distance i1 for the X-axis direction and distance j1 for the Y-axis direction as the tool moves toward the end point.

Designated by G18 or G19.

G17	XY plane
G18	ZX plane
G19	YZ plane

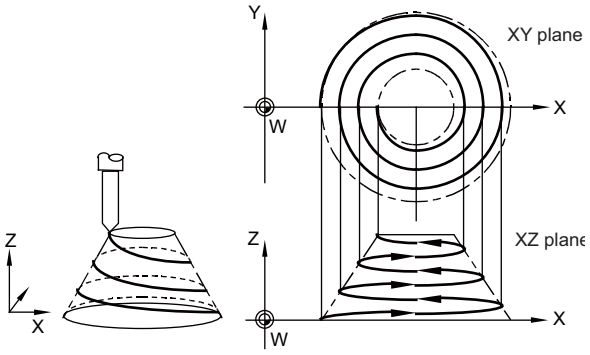
The direction of the arc rotation is designated by G02.1 or G03.1.

G02.1	Clockwise (CW)
G03.1	Counterclockwise (CCW)

The number of pitches (number of rotations) is designated by p1.
By assigning zero to p1, the pitch designation can be omitted in this case, the interpolation is obtained as a spiral rotation of less than one full turn. Assigning 1 to p1 yields a spiral rotation of more than one full turn but less than two full turns.

■ Conical interpolation

When an axis other than the ones for the spiral interpolation plane has been designated at the same time, the other axis will also be interpolated in synchronization with the spiral interpolation.
G17 G91 G02.1 X100. Z150. I150. P3 F500;
In the example given above, truncated cone interpolation is performed.



For command format type 2

■ Spiral interpolation

G02(G03) Xx1 Yy1 Ii1 Jj1 Qq1/Ll1 Ff1 ;	
G02,G03	: Arc rotation direction
Xx1,Yy1	: End point coordinate
Ii1,Jj1	: Arc center
Qq1	: Amount by which radius is incremented or decremented per spiral rotation
Ll1	: Number of pitches
Ff1	: Feed rate

- The number of spiral rotations when the increment/decrement amount of radius is specified can be calculated with the following expression.
  Number of rotations = |(arc end point radius - arc start point radius)| / |increment or decrement amount of radius|
- Q takes precedence if both Q and L have been designated at the same time.

■ Conical interpolation

G17 G02(G03) Xx1 Yy1 Zz1 Ii1 Jj1 Kk1 /Qq /Ll1 Ff1 ;	
G02,G03	: Arc rotation direction
Zz1	: End point coordinate in height direction
Ii1,Jj1	: Arc center
Kk1	: Amount by which height is incremented or decremented per spiral rotation
Qq1	: Amount by which radius is incremented or decremented per spiral rotation
Ll1	: Number of pitches
Ff1	: Feed rate

- The number of rotations when the height increment or decrement amount is specified can be calculated with the following expression.
  Number of rotations = |Height| / |Amount by which height is incremented or decremented (I,J,K)|
- Q takes precedence over K which in turn takes precedence over L if Q, K and L have been designated at the same time.
- The tolerable error range (absolute position) for when the commanded end point position is deviated from the end point position obtained from the number of pitches and increment/decrement amount is set with the parameters.

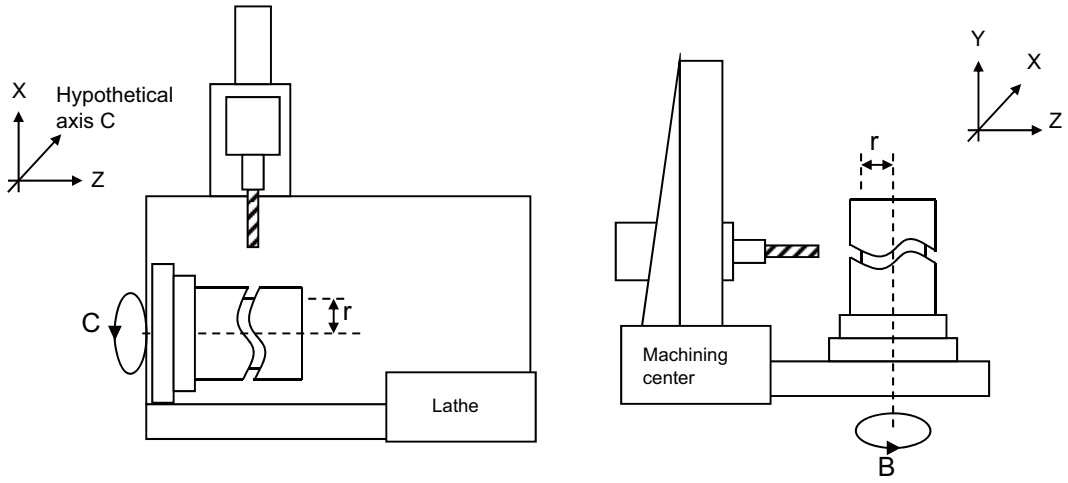
Common cautions for type 1 and 2

- This function cannot be used in combination with a tool radius compensation command (G41, G42).
- The arc plane always follows G17, G18 and G19. The plane arc control is carried out by G17, G18 and G19, even if designated by two addresses that do not match the plane.

3.2.5 Cylindrical Interpolation

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

This function transfers the shape that is on the cylinder's side surface (shape yielded by the cylindrical coordinate system) onto a plane, and when the transferred shape is designated in the program in the form of plane coordinates, the shape is converted into a movement along the linear and rotary axes of the original cylinder coordinates, and the contours are controlled by means of the CNC unit during machining.
Since the programming can be performed for the shapes produced by transferring the side surfaces of the cylinders, this function is useful when it comes to machining cylindrical cams and other such parts.
The command is programmed with the rotary axis and its orthogonal axis, which are different between the lathe and the machining center systems, to machine grooves and other shapes on the side of the cylinder.



■ Command format

Cylindrical interpolation mode start

G07.1 Name of rotary axis Cylinder radius;
or G107 Name of rotary axis Cylinder radius;

Name of rotary axis	: Axis name set to the rotary axis
Cylinder radius	: Radius value ≠ 0: Cylindrical interpolation mode start Command a value other than "0".

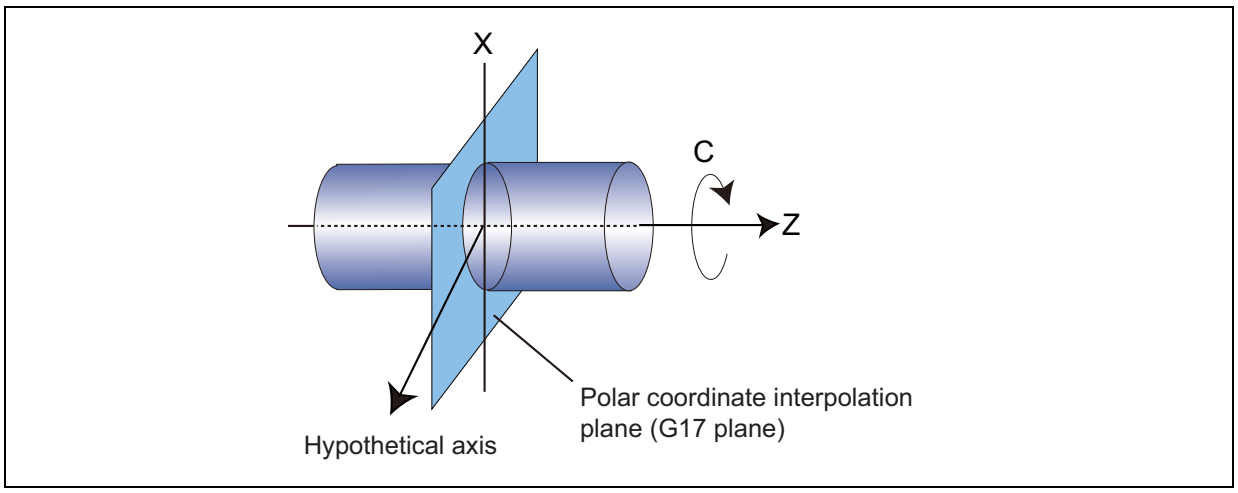
Cylindrical interpolation mode cancel

G07.1 Name of rotary axis 0 ;
or G107 Name of rotary axis 0;

3.2.6 Polar Coordinate Interpolation

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	—	—	—
L	Δ	Δ	Δ	Δ	○	○	○

This function converts the commands programmed by the orthogonal coordinate axes into linear axis movements (tool movements) and rotary axis movements (workpiece rotation) to control the contours. It is useful for cutting linear cutouts on the outside diameter of the workpiece, grinding cam shafts, etc.
This function can be used with the G code list 1, 6 and 7.



■ Polar coordinate interpolation mode

(G12.1)
The polar coordinate interpolation mode is established by designating the G12.1 command.
The plane that uses the linear axis as the plane's 1st axis, and the intersecting hypothetical axis as the plane's 2nd axis (hereafter "polar coordinate interpolation plane") is selected. Polar coordinate interpolation is performed on this plane. (Whether the linear axis is used as the plane's 1st axis or the plane's 2nd axis can be selected with the parameter.)

- Linear interpolation and circular interpolation can be designated in the polar coordinate interpolation mode.
- Either absolute command or incremental command can be issued.
- Tool radius compensation can be applied to the program commands.
  Polar coordinate interpolation is performed for the path after tool radius compensation.
- For the feed rate, designate a tangential rate on the polar coordinate interpolation plane (orthogonal coordinate system) using the F command.
  The F rate is in either mm/min or inch/mm units.

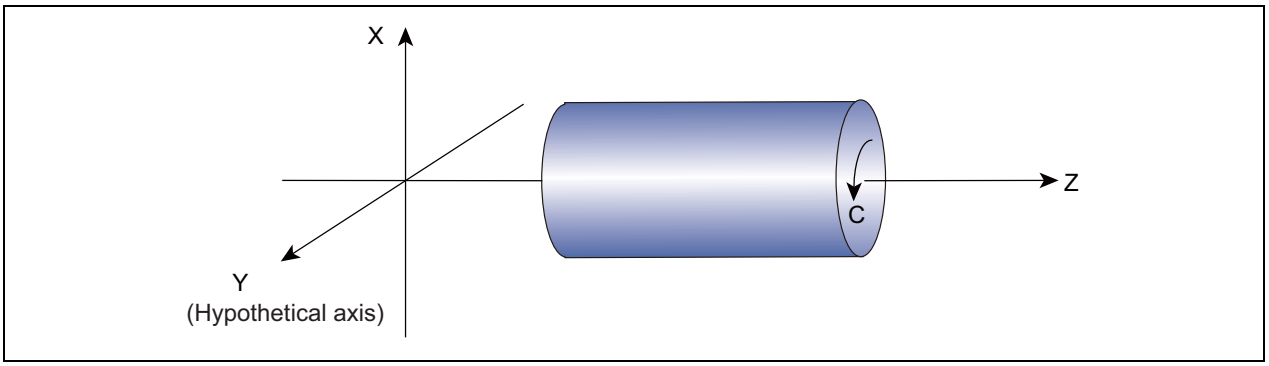
■ Polar coordinate interpolation cancel mode

(G13.1)
The polar coordinate interpolation cancel mode is established by designating the G13.1 command.

3.2.7 Milling Interpolation

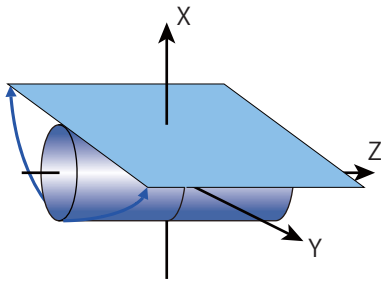
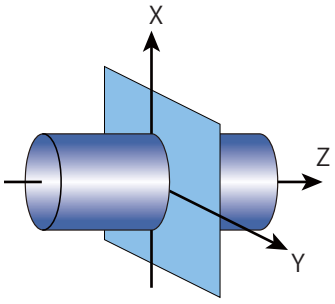
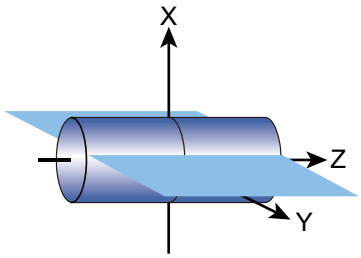
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	Δ	Δ	Δ	Δ	○	○	○

When a lathe with linear axes (X, Z axes) and rotary axis (C axis) serving as the control axes is to perform milling at a workpiece end face or in the longitudinal direction of the workpiece, this function uses the hypothetical axis Y which is at right angles to both the X and Z axes to enables the milling shape to be programmed as the X, Y and Z orthogonal coordinate system commands.
With this function, the workpiece can be treated as a cylinder with radius X, and commands can be designated on the plane formed by transferring the cylinder side surface instead.
With milling interpolation, the commands programmed by the orthogonal coordinate system are converted into linear axis and rotary axis movements (workpiece rotation) to control the contours.



G12.1 ;	**Milling mode ON**

G13.1 ;	**Milling mode OFF (Turning mode)**

G16(Y-Z cylindrical plane)	G17(X-Y plane)	G19(Y-Z plane)
		
Plane on which radius X cylinder is developed. Select this to machine the cylindrical plane of a workpiece.	X-Y plane in XYZ orthogonal coordinate system. Select this to machine the workpiece end face.	Y-Z plane in XYZ orthogonal coordinate system. Select this to machine a plane of a cylinder cut in the longitudinal direction.

3.2.8 Hypothetical Axis Interpolation

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	—	—	—
L	—	—	—	—	—	—	—

Take one of the axes of the helical interpolation or spiral interpolation, including a linear axis, as a hypothetical axis (axis with no actual movement) and perform pulse distribution. With this procedure, an interpolation equivalent to the helical interpolation or spiral interpolation looked from the side (hypothetical axis), or SIN or COS interpolation, will be possible.
The setting of this hypothetical axis is commanded with G07.

G07 Y0 ;	**X axis command cancel ON**
G07 Y1 ;	**X axis command cancel OFF**

G07	Hypothetical axis interpolation command
Y	Designate the axis for which hypothetical axis interpolation is performed. Designation of the axis for which axis command cancellation is performed applies for all the NC axes. (0: Cancel (normal), 1: Handle as hypothetical axis)

- Interpolation functions that are used for hypothetical interpolation are helical interpolation and spiral interpolation.
- During "G07α0;" to "G07α1;", α axis will be the hypothetical axis. Thus, when α axis is commanded independently during this time, dwell mode will be held until finishing the pulse distribution to the hypothetical axis.

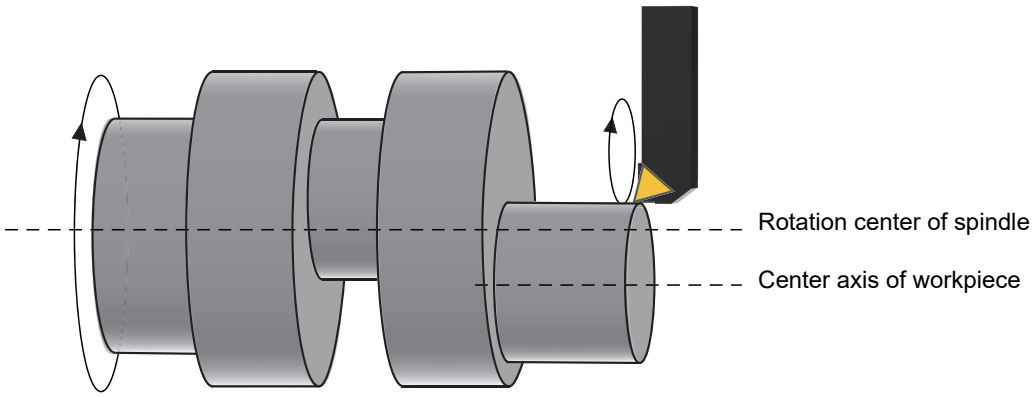
Note

(1) In order to perform hypothetical axis interpolation, helical interpolation must be added.

3.2.9 Eccentric Turning

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	△	△	△	△	○	○	—

This function modulates the radial position of the turning tool in synchronization with spindle rotation according to commands in a machining program. Using this function enables a workpiece whose center axis is offset from the spindle rotation center to be turned without re-clamping the workpiece.



■ Command format

Eccentric turning is a modal command of the group 1.
The rotary axis rotates clockwise (CW) for G02.6 and counterclockwise (CCW) for G03.6.
To cancel the eccentric turning command, issue another command of the group 1.
The eccentric turning must be commanded alone in a block.
During eccentric turning mode, do not issue commands for other functions at the same time as commands related to eccentric turning.

G17 G02.6 (G03.6) L__ R__ (Q__) X/U__ Y/V__ ,S__ (,I__) (,J__);
G18 G02.6 (G03.6) L__ R__ (Q__) Z/W__ X/U__ ,S__ (,I__) (,J__);
G19 G02.6 (G03.6) L__ R__ (Q__) Y/V__ Z/W__ ,S__ (,I__) (,J__);

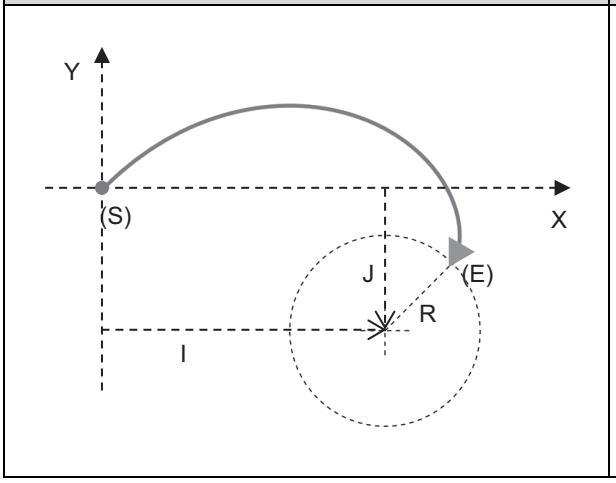
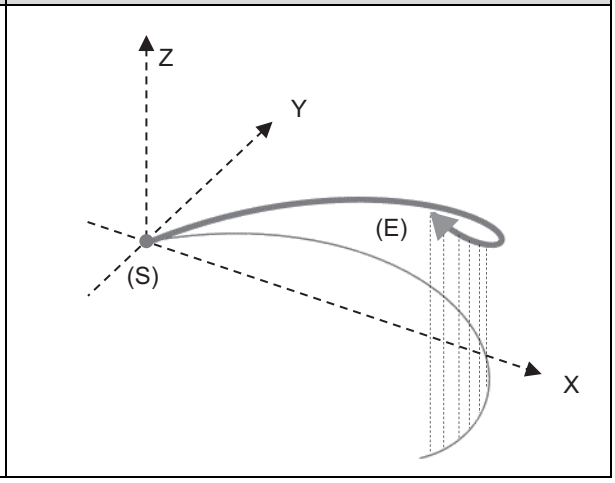
L	Number of workpiece rotations
R	Eccentric amount (initial value)
Q	Eccentric amount (end value)
Z/W	Block end point in longitudinal direction
X/U	Block end point in radial direction
,S	Spindle feedrate
,I	Major axis of eccentric elliptical workpiece
,J	Minor axis of eccentric elliptical workpiece

3.3 Curve Interpolation

3.3.1 Involute Interpolation

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	O	O	—
L	—	—	—	—	—	—	—

Involute interpolation moves a tool along an involute curve. This function can also move a tool helically (spirally) while moving the tool along the involute curve. This can be used for scroll machining of involute gears or compressors. The interpolation can eliminate a path step that is generated at a micro segment command, as well as acceleration/deceleration that is caused due to differences of segment length, leading to smooth accurate machining.
Using the automatic speed control function, including "involute interpolation override" and "acceleration clamping during involute interpolation", further improves the machining accuracy.

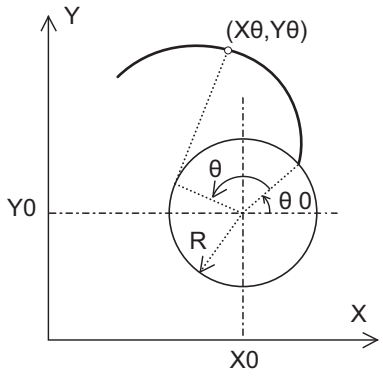
Involute Interpolation	Helical involute interpolation
	

(S) Start point
(E) End point

Program command path

Projected trajectory of XY plane of command program

Involute curve is obtained with the following expression:
X(θ) = R{cos(θ + θ0) + θ x sin(θ + θ0)} + X0
Y(θ) = R{sin(θ + θ0) - θ x cos(θ + θ0)} + Y0
Circle in the right figure is the base circle.



■ Command format

Involute Interpolation

G02.2 (G03.2) X__ Y__ I__ J__ R__ F__ ;	**G17 plane**

G02.2 (G03.2) Z__ X__ K__ I__ R__ F__ ;	**G18 plane**

G02.2 (G03.2) Y__ Z__ J__ K__ R__ F__ ;	**G19 plane**

G02.2 / G03.2	Involute curve rotation direction (G02.2: clockwise; G03.2: counterclockwise)
X	End point of involute interpolation (X axis)
Y	End point of involute interpolation (Y axis)
Z	End point of involute interpolation (Z axis)
I	Incremental value from the start point to the center of base circle (X axis)
J	Incremental value from the start point to the center of base circle (Y axis)
K	Incremental value from the start point to the center of base circle (Z axis)
R	Base circle radius
F	Feedrate (in involute curve tangent direction)

The range of command value follows the input range of coordinate position data.

Helical involute interpolation

G02.2 (G03.2) X__ Y__ α__ I__ J__ R__ F__ ;	**G17 plane**

G02.2 (G03.2) Z__ X__ α__ K__ I__ R__ F__ ;	**G18 plane**

G02.2 (G03.2) Y__ Z__ α__ J__ K__ R__ F__ ;	**G19 plane**

G02.2 / G03.2	Involute curve rotation direction (G02.2: clockwise; G03.2: counterclockwise)
X	End point of involute interpolation (X axis)
Y	End point of involute interpolation (Y axis)
Z	End point of involute interpolation (Z axis)
α	End point of linear axis (α: the name of the linear axis) When a rotary axis is designated, a program error occurs. Multiple linear axes can be designated.
I	Incremental value from the start point to the center of base circle (X axis)
J	Incremental value from the start point to the center of base circle (Y axis)
K	Incremental value from the start point to the center of base circle (Z axis)
R	Base circle radius
F	Feedrate (in involute curve tangent direction)

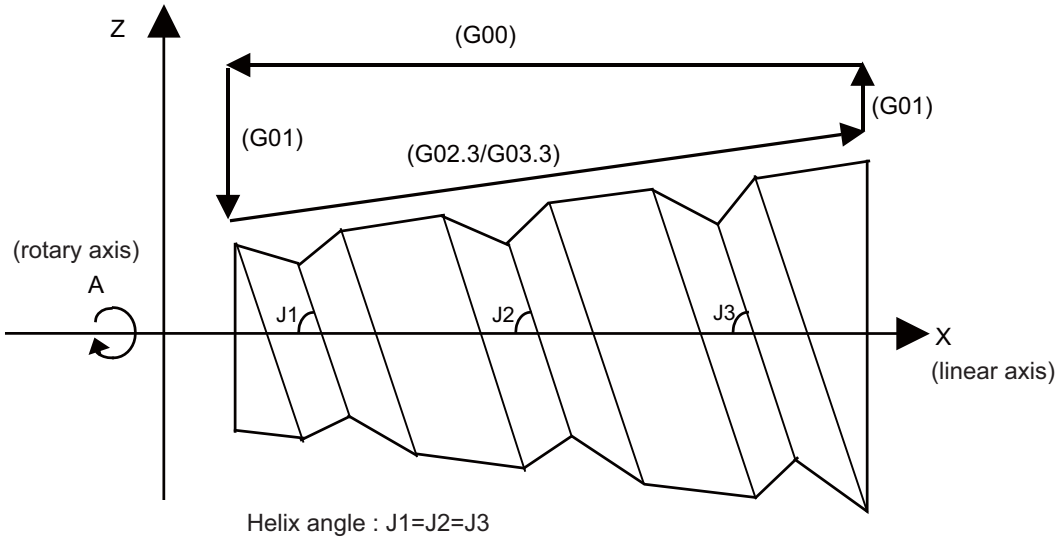
The range of command value follows the input range of coordinate position data.

3.3.2 Exponential Interpolation

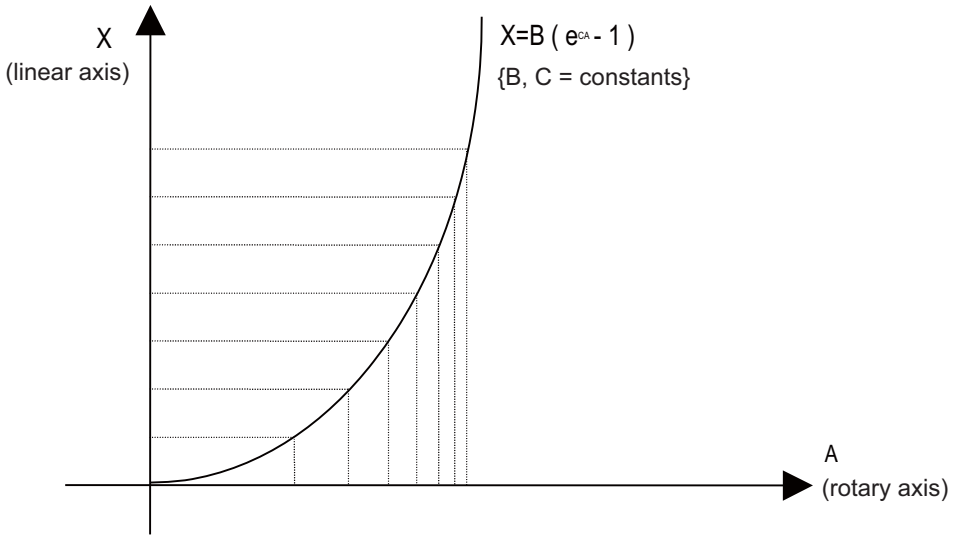
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	—	—	—
L	Δ	Δ	Δ	Δ	—	—	—

With this function, the rotary axis movement is changed into exponential functions vis-a-vis the linear axis movements. When exponential function interpolation is performed, linear interpolation is performed between the other axes and the linear axis. This makes it possible to machine tapered grooves (regular helix machining of tapered shapes) whose helix angle is always constant.
The function can be used for slotting and grinding end mills and other tools.

[Uniform helix machining of taper shape]



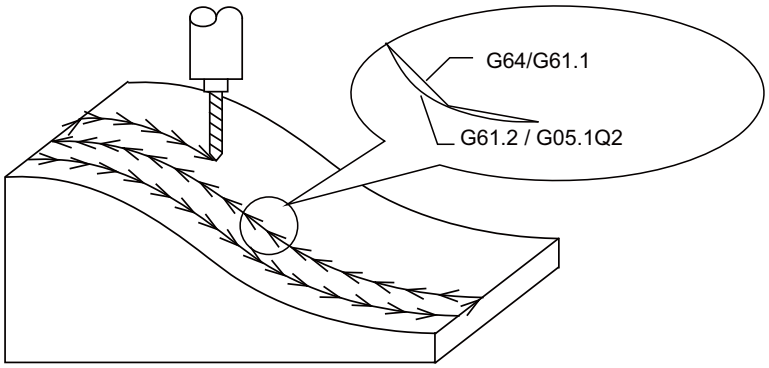
[Relationship between linear and rotary axes]



3.3.3 Spline Interpolation (G05.1Q2/G61.2)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	—
L	—	—	—	—	—	—	—

This function automatically generates spline curves that smoothly pass through rows of dots designated by a micro-segment machining program, and performs interpolation for the paths along the curves. This enables high-speed and high-accuracy machining to be achieved.
There are two types of the spline interpolation command format: G61.2 and G05.1Q2.
The high-speed high-accuracy control II/III is required for G05.1Q2 command. Command G05.1Q2 for the high-speed and smooth machining in the high-speed high-accuracy control II/III mode.



Note

(1) While the spline interpolation is valid, the high-speed high-accuracy control III operates as high-speed high-accuracy control II.

■ Command format

G61.2 command

G61.2 X__ Y__ Z__ F__ ; **or** **G61.2 ;**	**Spline interpolation mode ON**

G64 ; **.....**	**Spline interpolation mode OFF**

G05.1Q2 command

G05.1 Q2 X0 Y0 Z0 ;	**Spline interpolation mode ON**

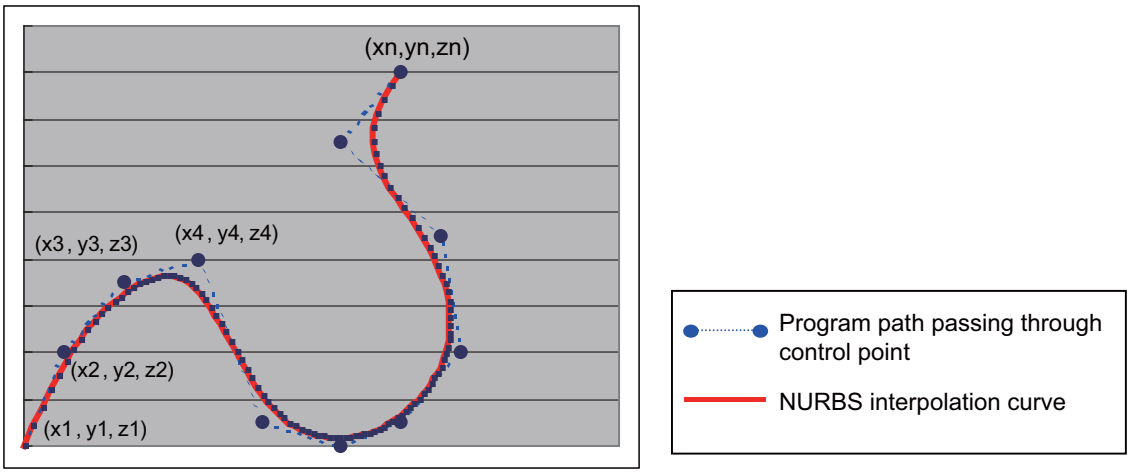
G05.1 Q0 ;	**Spline interpolation mode OFF**

3.3.4 NURBS Interpolation

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	—	—	—
L	—	—	—	—	—	—	—

This function realizes NURBS curve machining by commanding NURBS curve parameters (number of stages, weight, knot, control point). The path does not need to be replaced with micro segments.
This function operates only in the high-speed high-accuracy control II/III mode, so the high-speed high-accuracy control II/III function is required.
During NURBS interpolation, interpolation takes place at the commanded speed. However, if the curvature is large, the speed is clamped so that the machine's tolerable acceleration rate is not exceeded.

NURBS interpolation cannot be used during graphic check (continuous/step check).
Linear interpolation that connects the control points is used during graphic check.
The multi-part system simultaneous high-accuracy function is required when the NURBS interpolation is commanded to 2nd part system or higher.



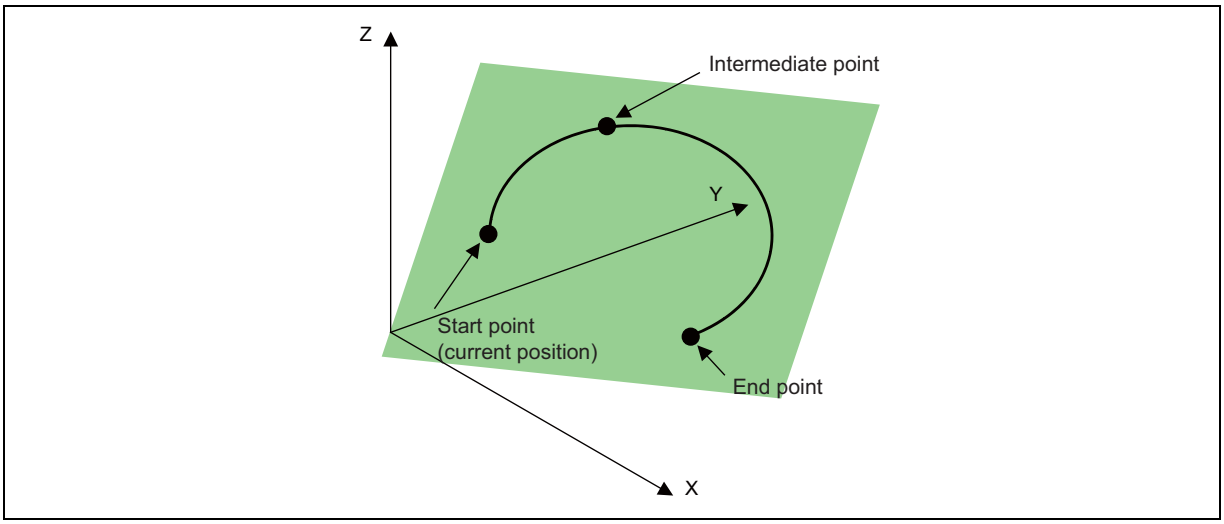
Note

(1) While the NURBS interpolation is valid, the high-speed high-accuracy control III operates as high-speed high-accuracy control II.

3.3.5 3-Dimensional Circular Interpolation

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	—
L	—	—	—	—	—	—	—

To issue a circular command over a three-dimensional space, an arbitrary point (intermediate point) must be designated on the arc in addition to the start point (current position) and end point. Using the 3-dimensional circular interpolation command, an arc shape determined by the three points (start point, intermediate point, end point) designated on the three-dimensional space can be machined.
The speed command during 3-dimensional circular interpolation is the tangential velocity on the arc.



■ Command format

G02.4 (G03.4) Xx1 Yy1 Zz1 αα1... ;	Intermediate point designation (1st block)
Xx2 Yy2 Zz2 αα2 ... ;	End point designation (2nd block)
G02.4 (G03.4)	: 3-dimensional circular interpolation command
Xx1, Yy1, Zz1	: Intermediate point coordinates
Xx2, Yy2, Zz2	: End point coordinates
αα1	: Arbitrary axis other than axis used as the reference in 3-dimensional circular interpolation (May be omitted)

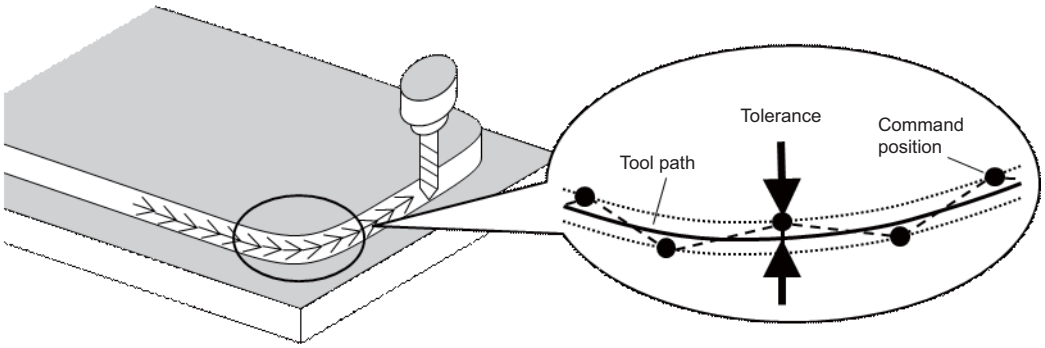
- The operation is the same for G02.4 and G03.4. (The rotation direction cannot be designated.)
- The axes used as the reference in 3-dimensional circular interpolation are the three basic axes set with the parameters.
- The X, Y, Z address in the block may be omitted. The intermediate point coordinates omitted in the 1st block become the start point coordinates, and the end point coordinates omitted in the 2nd block become the intermediate point coordinates.
- When using the 3-dimensional circular interpolation command, an arbitrary axis can be commanded in addition to the orthogonal coordinate system (X, Y, Z) used as the reference. The arbitrary axis designated in the intermediate point designating block (1st block) will interpolate to the command point when moving from the start point to intermediate point movement. The arbitrary axis designated in the end point command block (2nd block) will interpolate to the command point when moving from the intermediate point to the end point. The number of arbitrary axes that can be commanded differs according to the number of simultaneous contour control axes. The total of the basic three axes used as the reference of the 3-dimensional circular interpolation and the arbitrary axes commanded simultaneously must be less than the number of simultaneous contour control axes.
- To command 3-dimensional circular interpolation when the incremental command is enabled, specify as follows.
  Intermediate point coordinates: The position of the intermediate point relative to the start point
  End point coordinates: The position of the end point relative to the intermediate point

3.3.6 Spline Interpolation 2 (G61.4)

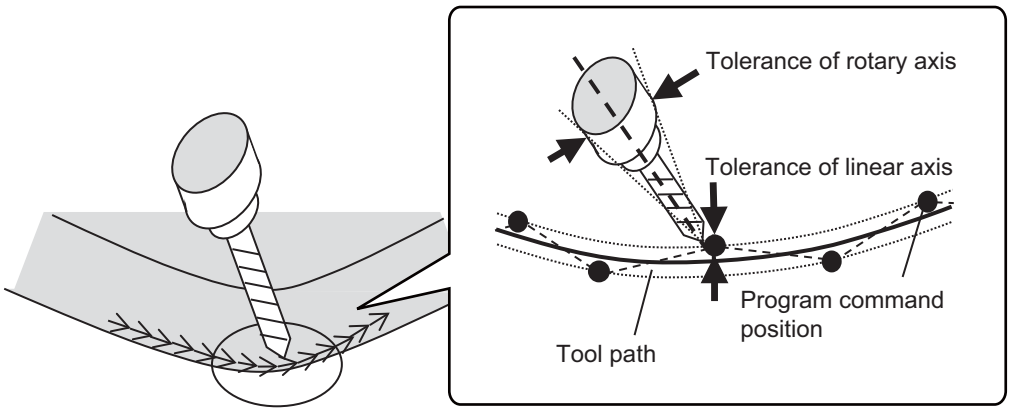
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	—
L	—	—	—	—	—	—	—

This function automatically generates a curve that smoothly passes within the tolerance range and interpolates the path along this curve. By using this function, smooth machining can be achieved even when the speed commanded in the program is low or the program has minute steps.
This function allows the machine to operate with the optimum tool path and speed, simply by specifying the tolerance size, so an operator can easily attain high quality machining.
This function is enabled under the tolerance control, so the specification of the tolerance control is required, too.

The tolerance size is the allowable error value between the path commanded by machining program and the path output with NC.



When spline interpolation 2 is used in combination with tool center point (TCP) control, spline interpolation 2 is performed with 5 axes. It generates a curve that passes through the TCP points smoothly within the tolerance, with the rotary axis angle also within the tolerance. The tool moves along the curve.



■ Command format

G61.4 (,K__) (,R__) ;	**Spline interpolation 2 mode ON**
,K	Tolerance size (mm) linear axis
,R	Tolerance size (deg) rotary axis

Spline interpolation 2 mode with command G61.4 will be cancelled by designating any one of G code group 13.

- G61 (Exact stop check mode)
- G61.1 (High-accuracy control mode)
- G61.2 (Spline interpolation command)
- G62 (Automatic corner override)
- G63 (Tapping mode)
- G64 (Cutting mode)
- G08P1 (High-accuracy control mode start)
- G08P0 (High-accuracy control mode end)

4

Feed

4.1 Feedrate

4.1.1 Rapid Traverse Rate (m/min)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	1000	1000	1000	1000	1000	1000	1000
L	1000	1000	1000	1000	1000	1000	1000

The rapid traverse rate is set in the parameters for each axis.
During high-accuracy control mode, the dedicated rapid traverse rate (parameter) is applied.
Override can be applied to the rapid traverse rate using the external signal supplied.

The rapid traverse rate is effective for the following commands.
M system: G00, G27, G28, G29, G30, G60
L system: G00, G27, G28, G29, G30

The high-accuracy control mode rapid traverse rate is effective for the following commands.
M system: High-accuracy control, high-speed high-accuracy control I/II/III, SSS control, high-accuracy spline control
L system: High-accuracy control, high-speed high-accuracy control I/II, SSS control

Rapid traverse rate and high-accuracy control mode rapid traverse rate setting (*1)

Least command increment	B	C	D	E
Metric input (mm/min, °/min)	1 to 1000000	1 to 1000000	1 to 1000000	1 to 1000000
Inch input (inch/min)	1 to 100000	1 to 100000	1 to 100000	1 to 100000

Least command increment B: 0.001 mm (0.0001 inch)
Least command increment C: 0.0001 mm (0.00001 inch)
Least command increment D: 0.00001 mm (0.000001 inch)
Least command increment E: 0.000001 mm (0.0000001 inch)

(*1) For spindle position control (spindle/C axis control), the upper limit of rapid traverse rate is as follows.
Metric input (mm/min, °/min): 2000000
Inch input (inch/min): 200000

4.1.2 Cutting Feedrate (m/min)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	1000	1000	1000	1000	1000	1000	1000
L	1000	1000	1000	1000	1000	1000	1000

[M system]

This function specifies the feedrate of the cutting commands, and a feed amount per spindle rotation or feed amount per minute is commanded.

Once commanded, it is stored in the memory as a modal value. The feedrate modal value is cleared to zero only when the power is turned ON.

The maximum cutting feedrate is clamped by the cutting feedrate clamp parameter (whose setting range is the same as that for the cutting feedrate).

If the high-accuracy control mode's cutting clamp speed is set, the cutting feedrate will be clamped at that speed during high-accuracy control, high-speed high-accuracy control, high-accuracy spline control or SSS control.

- If the value set for high-accuracy control mode cutting clamp speed is 0, the axis will be clamped at the cutting feed clamp speed.
- High-accuracy control mode cutting clamp speed is set with the parameters.

Cutting feed clamp speed setting range (*1)

Least command increment	B	C	D	E
Metric input (mm/min, °/min)	0.001 to 1000000	0.0001 to 1000000	0.00001 to 1000000	0.000001 to 1000000
Inch input (inch/min)	0.0001 to 100000	0.00001 to 100000	0.000001 to 100000	0.0000001 to 100000

Least command increment B: 0.001 mm (0.0001 inch)
Least command increment C: 0.0001 mm (0.00001 inch)
Least command increment D: 0.00001 mm (0.000001 inch)
Least command increment E: 0.000001 mm (0.0000001 inch)

(*1) For spindle position control (spindle/C axis control), the upper limit of cutting feed clamp speed is as follows.
Metric input (mm/min, °/min): 2000000
Inch input (inch/min): 200000

<G code command for which the cutting feedrate is effective>
For others such as G01,G02,G03,G02.1,G03.1,G33, etc., refer to the interpolation specifications.

[L system]

This function specifies the feedrate of the cutting commands, and a feed amount per spindle rotation or feed amount per minute is commanded.

Once commanded, it is stored in the memory as a modal value. The feedrate modal value is cleared to zero only when the power is turned ON.

The maximum cutting feedrate is clamped by the cutting feedrate clamp parameter (whose setting range is the same as that for the cutting feedrate).

Cutting feed clamp speed setting range (*1)

Least command increment	B	C	D	E
Metric input (mm/min, °/min)	0.001 to 1000000	0.0001 to 1000000	0.00001 to 1000000	0.000001 to 1000000
Inch input (inch/min)	0.0001 to 100000	0.00001 to 100000	0.000001 to 100000	0.0000001 to 100000

Least command increment B: 0.001 mm (0.0001 inch)
Least command increment C: 0.0001 mm (0.00001 inch)
Least command increment D: 0.00001 mm (0.000001 inch)
Least command increment E: 0.000001 mm (0.0000001 inch)

(*1) For spindle position control (spindle/C axis control), the upper limit of cutting feed clamp speed is as follows.
Metric input (mm/min, °/min): 2000000
Inch input (inch/min): 200000

<G code command for which the cutting feedrate is effective>
For others such as G01,G02,G03,G02.1,G03.1,G33, etc., refer to the interpolation specifications.

4.1.3 Manual Feedrate (m/min)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	1000	1000	1000	1000	1000	1000	1000
L	1000	1000	1000	1000	1000	1000	1000

The manual feedrate are designated as the feedrate in the jog mode or incremental feed mode for manual operation and the feedrate during dry run ON for automatic operation. The manual feedrate are set with external signals.
The "manual feedrate" signals from the PLC include two methods, the code method and value setting method.
Which method to be applied is determined with a signal common to the entire system. The signals used by these methods are common to all axes.

- Setting range under the code method

  Metric input 0.00 to 14000.00 mm/min (31 steps)
  Inch input 0.000 to 551.000 inch/min (31 steps)

- Setting range under the value setting method

  Metric input 0 to 1000000.00 mm/min in 0.01 mm/min increments
  Inch input 0 to 39370 inch/min in 0.001 inch/min increments

  Multiplication factor PCF1 and PCF2 are available with the value setting method.

4.1.4 Rotary Axis Command Speed Tenfold

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

This function multiplies the rotary axis' command speed by 10 during initial inching.
The commanded speeds are as follow.

Automatic operation	**Cutting feedrate**	For the inch system, the rotary axis command speed is multiplied by 10. For example, if the B axis is the rotary axis in the inch system and the following type of machining program is executed, the rotary axis command speed will be multiplied by 10, and the rotary axis will move at 1000 deg./min. N1 G1 B100. F100.;
	Rapid traverse rate	The rapid traverse rate is not multiplied by 10, and is the speed set in the parameters.
Manual operation		The command speeds related to manual operation, such as JOG feed, are not multiplied by 10. The display speed unit also remains as "deg./min".

4.2 Feedrate Input Methods

4.2.1 Feed per Minute (Asynchronous Feed)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

[M system]
By issuing the G94 command, the commands from that block are issued directly by the numerical value following F as the feedrate per minute (mm/min, inch/min).

Metric input

Input setting increment	B (0.001 mm)	C (0.0001 mm)	D (0.00001 mm)	E (0.000001 mm)
Command address	F (mm/min)	F (mm/min)	F (mm/min)	F (mm/min)
Least command increment	1 (= 1.000) (1. = 1.000)	1 (= 1.0000) (1. = 1.0000)	1 (= 1.00000) (1. = 1.00000)	1 (= 1.000000) (1. = 1.000000)
Command range	0.001 - 1000000.000	0.0001 - 1000000.0000	0.00001 - 1000000.00000	0.000001 - 1000000.000000

Inch input

Input setting increment	B (0.0001 inch)	C (0.00001 inch)	D (0.000001 inch)	E (0.0000001 inch)
Command address	F (inch/min)	F (inch/min)	F (inch/min)	F (inch/min)
Least command increment	1 (= 1.0000) (1. = 1.0000)	1 (= 1.00000) (1. = 1.00000)	1 (= 1.000000) (1. = 1.000000)	1 (= 1.0000000) (1. = 1.0000000)
Command range	0.0001 - 100000.0000	0.00001 - 100000.00000	0.000001 - 100000.000000	0.0000001 - 100000.0000000

[L system]
By issuing the G94 command, the commands from that block are issued directly by the numerical value following F as the feedrate per minute (mm/min, inch/min).

Metric input

Input setting increment	B (0.001 mm)	C (0.0001 mm)	D (0.00001 mm)	E (0.000001 mm)
Command address	F (mm/min)	F (mm/min)	F (mm/min)	F (mm/min)
Least command increment	1 (= 1.000) (1. = 1.000)	1 (= 1.0000) (1. = 1.0000)	1 (= 1.00000) (1. = 1.00000)	1 (= 1.000000) (1. = 1.000000)
Command range	0.001 - 1000000.000	0.0001 - 1000000.0000	0.00001 - 1000000.00000	0.000001 - 1000000.000000

Inch input

Input setting increment	B (0.0001 inch)	C (0.00001 inch)	D (0.000001 inch)	E (0.0000001 inch)
Command address	F (inch/min)	F (inch/min)	F (inch/min)	F (inch/min)
Least command increment	1 (= 0.0100) (1. = 1.0000)	1 (= 0.01000) (1. = 1.00000)	1 (= 0.010000) (1. = 1.000000)	1 (= 0.0100000) (1. = 1.0000000)
Command range	0.0001 - 100000.0000	0.00001 - 100000.00000	0.000001 - 100000.000000	0.0000001 - 100000.0000000

4.2.2 Feed per Revolution (Synchronous Feed)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	○	○	○	○	○	○	○

By issuing the G95 command, the commands from that block are issued directly by the numerical value following F as the feedrate per spindle revolution (mm/revolution or inch/revolution).
The least command increment and command range of the feedrate designation F are as follows.

[M system]
Metric input

Input setting increment	B (0.001 mm)	C (0.0001 mm)	D (0.00001 mm)	E (0.000001 mm)
Command address	F (mm/rev)	F (mm/rev)	F (mm/rev)	F (mm/rev)
Least command increment	1 (= 0.001) (1. = 1.000)	1 (= 0.0001) (1. = 1.0000)	1 (= 0.00001) (1. = 1.00000)	1 (= 0.000001) (1. = 1.000000)
Command range	0.001 - 999.999	0.0001 - 999.9999	0.00001 - 999.99999	0.000001 - 999.999999

Inch input

Input setting increment	B (0.0001 inch)	C (0.00001 inch)	D (0.000001 inch)	E (0.0000001 inch)
Command address	F (inch/rev)	F (inch/rev)	F (inch/rev)	F (inch/rev)
Least command increment	1 (= 0.0001) (1. = 1.0000)	1 (= 0.00001) (1. = 1.00000)	1 (= 0.000001) (1. = 1.000000)	1 (= 0.0000001) (1. = 1.0000000)
Command range	0.0001 - 999.9999	0.00001 - 999.99999	0.000001 - 999.999999	0.0000001 - 999.9999999

[L system]
Metric input

Input setting increment	B (0.001 mm)	C (0.0001 mm)	D (0.00001 mm)	E (0.000001 mm)
Command address	F (mm/rev)	F (mm/rev)	F (mm/rev)	F (mm/rev)
Least command increment	1 (= 0.0001) (1. = 1.00)	1 (= 0.00001) (1. = 1.00)	1 (= 0.000001) (1. = 1.00)	1 (= 0.0000001) (1. = 1.00)
Command range	0.0001 - 999.9999	0.00001 - 999.99999	0.000001 - 999.999999	0.0000001 - 999.9999999

Inch input

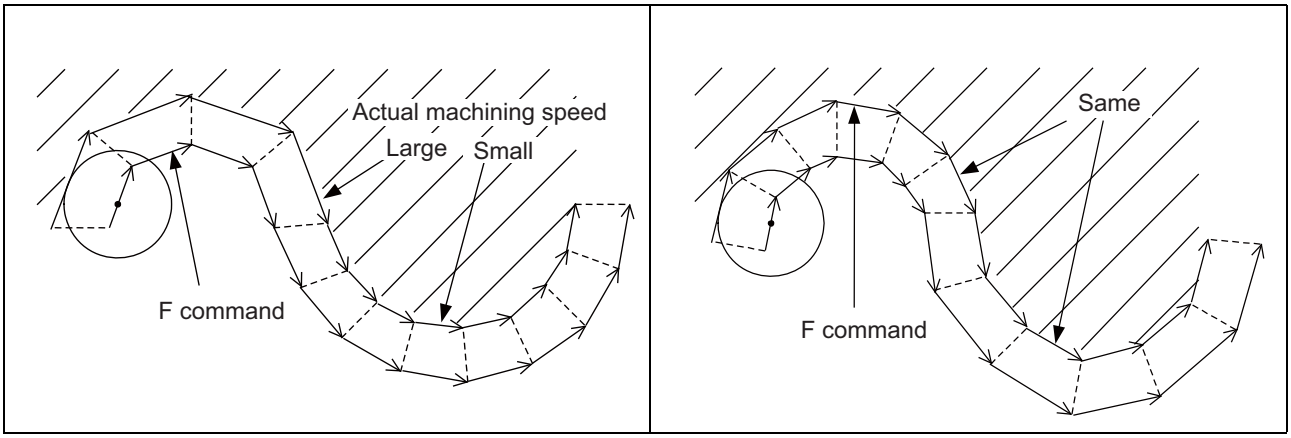
Input setting increment	B (0.0001 inch)	C (0.00001 inch)	D (0.000001 inch)	E (0.0000001 inch)
Command address	F (inch/rev)	F (inch/rev)	F (inch/rev)	F (inch/rev)
Least command increment	1 (= 0.000001) (1. = 1.000)	1 (= 0.0000001) (1. = 1.000)	1 (= 0.00000001) (1. = 1.000)	1 (= 0.000000001) (1. = 1.000)
Command range	0.00001 - 99.99999	0.000001 - 99.999999	0.0000001 - 99.9999999	0.00000001 - 99.99999999

4.2.3 Inverse Time Feed

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	—
L	—	—	—	—	—	—	—

During inside cutting when machining curved shapes with tool radius compensation applied, the machining speed on the cutting surface becomes faster than the tool center feedrate. Therefore, problems such as reduced accuracy on the cutting surface may occur.
This reduced accuracy can be prevented with inverse time feed. This function can, in place of normal feed commands, issue one block of machining time (inverse) in F commands. The machining speed on the cutting surface is constantly controlled, even if radius compensation is applied to the machining program that expresses the free curve surface with micro segment lines.
Note that when the calculated machining time exceeds the cutting feed clamp speed, the F command value in the inverse time feed follows the cutting feed clamp speed.

Regular F command **Inverse time feed**



■ Command format

G93 ;	**Inverse Time Feed**

Inverse time feed (G93) is a modal command and is valid until feed per minute (G94) or feed per revolution (G95) is commanded.

G00 Xx1 Yy1 ;	
G93 ;	-> Inverse time feed mode ON
G01 Xx2 Yy2 Ff2 ;	-> In inverse time feed mode
G02 Xx3 Yy3 Ii3 Jj3 Ff3 ;	:
G94(G95) ;	-> Inverse time feed mode OFF

In movement blocks, since processing time is commanded to a line segment, command the feedrate "F" each time.

4.2.4 F 1-Digit Feed

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

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 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 an F 1-digit command has been issued, the external output signal is output.

When the programmed feedrate has been issued as an F 1-digit command, the feedrate can be increased or reduced by turning the manual handle.
The feedrate cannot be changed by the 2nd and 3rd handles.
The F 1-digit feed speed changed by manual handle can be retained after power is turned OFF by setting parameters.

■ Amount by which speed is varied by manual handle

Speed variation amount ΔF is expressed by the equation below:

$$\Delta F = \Delta P \times \frac{FM}{K}$$

ΔP : Handle pulse (±)
FM : F1 to F5 upper limit (parameter setting)
K :Speed variation constant (parameter setting)

(Example) When the feedrate is to be increased or reduced by 10 mm/min per manual handle scale increment
If FM is made 3600 mm/min, "K" must be set to "360" according to the following formula.

$$\Delta F = 10 = 1 \times \frac{3600}{K}$$

■ Enabling conditions

- Cycle start must be underway.
- Cutting feed must be underway, and the F 1-digit feedrate must be designated.
- The F 1-digit valid parameter must be ON
- The "F 1-digit speed change valid" signal must be ON.
- The handle mode must not be selected.
- A dry run must not be in progress.
- Machine lock must not be activated.
- The parameter settings (the F 1-digit feedrate upper limit and F 1-digit feedrate change constant) must not be "0".

4.2.5 Manual Speed Command

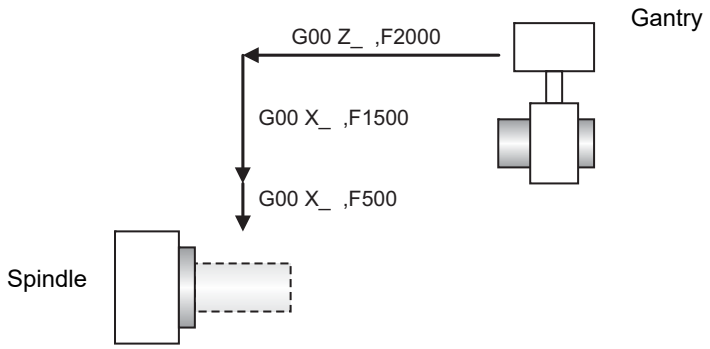
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

In the memory or MDI mode, validate the manual speed command and select either handle feed or jog (manual) feed so that the automatic operation is carried out at the feedrate.
With a command in the (-) direction, the program path can be reversed. Note that, however, program path can be reversed only within the currently executing block and not beyond the block.
Whether or not to execute reverse run with a command in the (-) direction is set with the PLC interface.
Furthermore, by setting the parameter, handle, jog and manual feed speed can be executed at the feed speed according to the ratio of program command speed of running block when issuing the manual speed command in multiple systems.

4.2.7 G00 Feedrate Designation (,F Command)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

Feedrates can be specified for G00 (positioning command).
The speed of tool exchange, axis movement of gantry, etc. can be specified with the machining program so that the mechanical vibration can be suppressed.
Operations other than the feedrate follows the G00 specification.



■ Command format

The feedrates of the movement by the G00 block, the movement during the G00 mode, and the movement during the fixed cycle for drilling are commanded with ",F".
Feedrate designation in G00 block

```
G00 X__ Z__ ,F1000 ;
```

Feedrate designation for movement command in G00 mode

```
G00 ;
X__ Z__ ,F1000 ;
```

Note

(1) , F command is ignored in the G01 mode.

Feedrate designation for movement command during drilling cycle

```
G8 □ (G7 □ ) X/Z__ ..... ,F1000;
X/Z__ ..... ,F500;
 :
G80
 :
```

L system	G83/G87: Deep-hole drilling, G84/G88: Tapping, G85/G89: Boring
M system	G81: Drilling/Spot drilling, G82: Drilling/Counter boring, G83: Deep-hole drilling, G84: Tapping, G85/G86: Boring, G87: Back boring, G88/G89: Boring, G73: Step, G74: Reverse tapping, G75: Circular cutting, G76: Fine boring

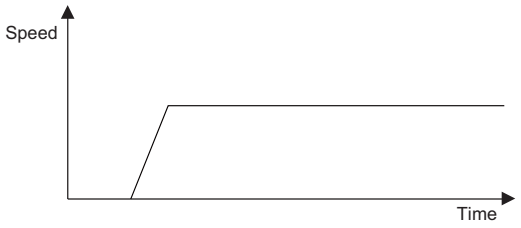
,F command range

- The range is equal to the range of the feed per minute F command (mm/min, inch/min) in the G01 mode.
- Commands that exceed the command range cause the error "P67: F value is exceeding the limit".
- Switching inch/mm is invalid for rotary axes.

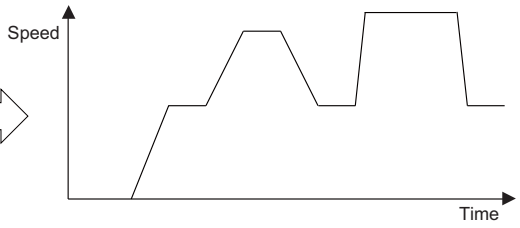
4.2.8 Selection of Axis (Axes) for Feedrate Command

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	—	—	—
L	—	—	—	—	—	—	—

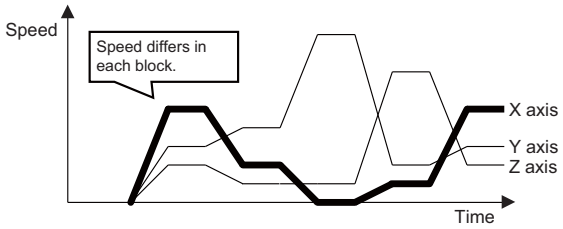
This function enables designation of the feedrate of a specific axis (with the command F).
When the feedrate of a specific axis fluctuates wildly, desired surface finish may fail to be attained. The fluctuation can be suppressed with this function, resulting in improved surface quality.



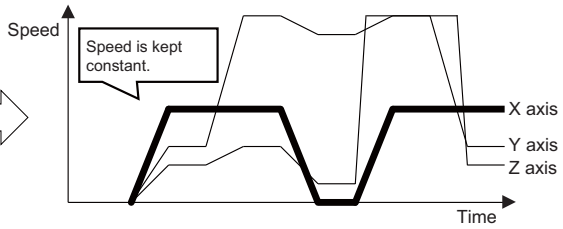
Resultant speed of all axes when the function is invalid



Resultant speed of all axes when the function is valid
(with X axis being selected)



Feedrate of each axis when the function is invalid



Feedrate of each axis when the function is valid
(with X axis being selected)

■ Command format

Selection of Axis (Axes) for Feedrate Command

G130 Axis name 1 Axis name 2 ... Axis name n ;
Axis name n : Axis (axes) for feedrate command as axis address

Selection of axis (axes) for feedrate command cancel

G130 ;

- G130 is a non-modal command of group 0.
- G130 must be commanded in an independent block. A program error occurs when another G code is commanded in the same block.
- Designate the axis name "n" using an axis address set in the parameter. If the designated axis does not exist, a program error occurs.
- When an axis address enabled with the axis name extension function is specified for the axis name "n", a program error occurs.
- The selection of axis (axes) for feedrate command is canceled with reset 1, reset 2, Reset & Rewind, or emergency stop.

4.3 Override

4.3.1 Rapid Traverse Override

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

■ Type 1 (code method)

Four levels of override (1%, 25%, 50% and 100%) can be applied to manual or automatic rapid traverse using the external input signal supplied.

■ Type 2 (value setting method)

Override can be applied in 1% steps from 0% to 100% to manual or automatic rapid traverse using the external input signal supplied.

Note

(1) Type 1 and type 2 can be selected by PLC processing.
(2) This function requires a built-in PLC for type 2.

4.3.2 Cutting Feed Override

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

■ Type 1 (code method)

Override can be applied in 10% steps from 0% to 300% to the feed rate command designated in the machining program using the external input signal supplied.

■ Type 2 (value setting method)

Override can be applied in 1% steps from 0% to 327% to the feed rate command designated in the machining program using the external input signal supplied.

Note

(1) This function requires a built-in PLC for type 2.

4.3.3 2nd Cutting Feed Override

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

Override can be further applied in 0.01% steps from 0% to 327.67% as a second stage override to the feed rate after the cutting feed override has been applied.

Note

(1) This function requires a built-in PLC.

4.3.4 Override Cancel

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

By turning on the "Override cancel external" signal, the override is automatically set to 100% for the cutting feed during an automatic operation mode (tape, memory and MDI).

Note

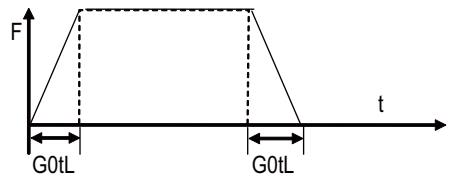
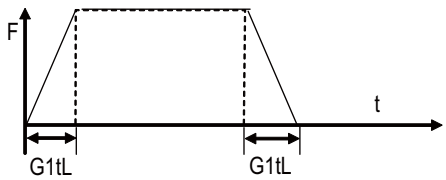
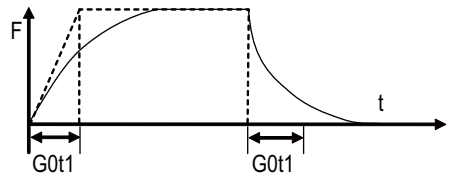
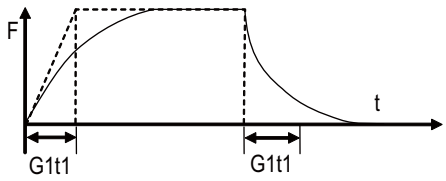
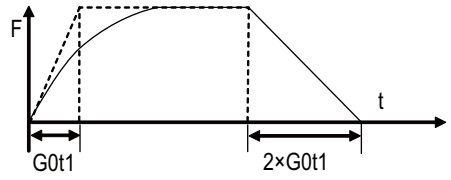
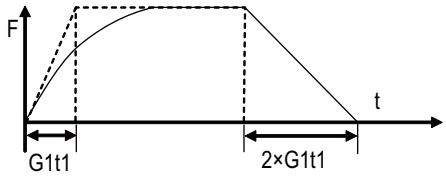
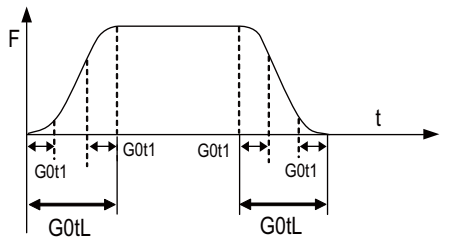
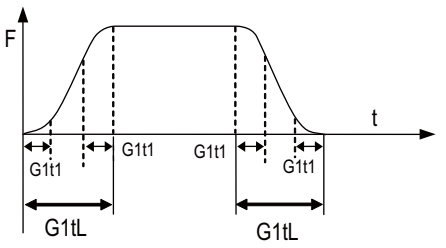
(1) The "Override cancel" signal is not valid for manual operation.
(2) When the cutting feed override or 2nd cutting feed override is 0%, the 0% override takes precedence and the override is not canceled.
(3) The "Override cancel" signal is not valid for rapid traverse.

4.4 Acceleration/Deceleration

4.4.1 Automatic Acceleration/Deceleration After Interpolation

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

Acceleration/deceleration is applied to all commands automatically. The acceleration/deceleration patterns are linear acceleration/deceleration, soft acceleration/deceleration, exponent function acceleration/ deceleration, exponent function acceleration/linear deceleration and any of which can be selected by using a parameter.
For rapid traverse feed or manual feed, acceleration/deceleration is always made for each block, and the time constant can be set for each axis separately.

Rapid traverse	Cutting feed
<Linear acceleration/deceleration> 	<Linear acceleration/deceleration> 
<Primary delay> 	<Primary delay> 
<Exponential acceleration-Linear deceleration> 	<Exponential acceleration-Linear deceleration> 
<Soft acceleration/deceleration> 	<Soft acceleration/deceleration> 

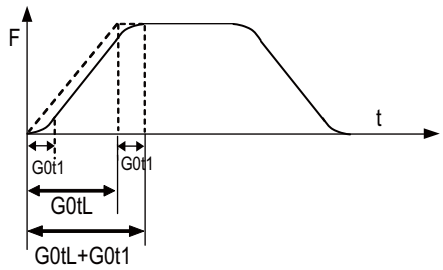
G0tL: G0 time constant (linear)
G0t1: G0 time constant (primary delay)/The 2nd step time constant of the soft acceleration/deceleration
G1tL: G1 time constant (linear)
G1t1: G1 time constant (primary delay)/The 2nd step time constant of the soft acceleration/deceleration

Note

(1) The rapid traverse feed acceleration/deceleration patterns are effective for the following:
G00, G27, G28, G29, G30, rapid traverse feed in manual run, JOG feed, incremental feed, return to reference position.
Invalid for G31 and handle feed.

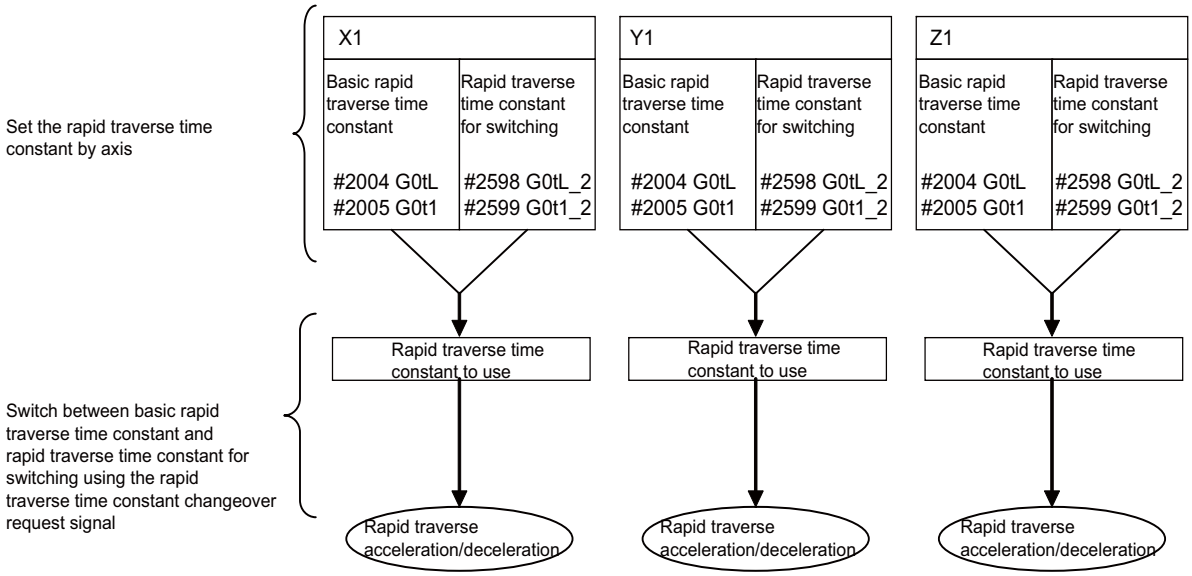
(2) Acceleration/deceleration in handle feed mode is usually performed according to the acceleration/deceleration pattern for cutting feed. However, a parameter can be specified to select a pattern with no acceleration/deceleration (step).

(3) Acceleration/deceleration time of soft acceleration/deceleration can be changed by parameter as follows.



Total accelerating time:G0tL+G0t1
S-pattern part time:G0t1
Linear part time:G0tL-G0t1

A rapid traverse time constant can be switched by "Rapid traverse time constant: Switchover request" signal.



4.4.2 Rapid Traverse Constant-Gradient Acceleration/Deceleration

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

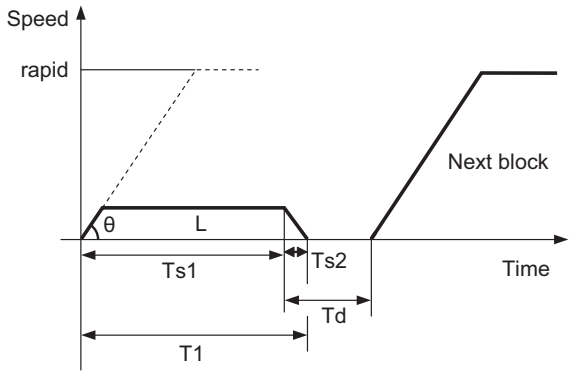
This function performs acceleration and deceleration at a constant-gradient during linear acceleration/deceleration by the rapid traverse command or the linear interpolation command. Compared to the method of acceleration/deceleration after interpolation, the constant-gradient acceleration/deceleration method makes for improved cycle time.
This function is valid for the rapid traverse (G00) command and the linear interpolation (G01) command only when the acceleration/deceleration mode is set to linear acceleration/deceleration or soft acceleration/deceleration.

(Example) When the interpolation distance is so short that the rapid traverse rate is not achieved

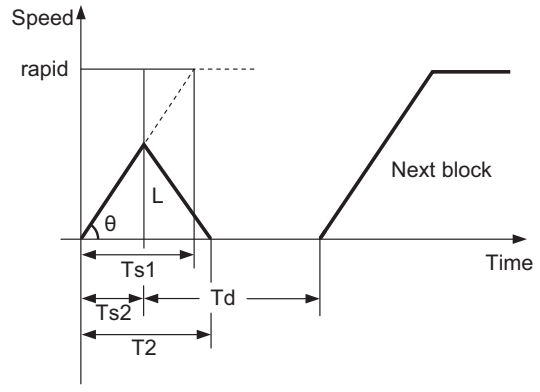
For acceleration/deceleration with fixed time constant, the gradient is determined by the rapid traverse rate; however, the cycle time is determined by the time constant, and the reaching speed is slower than constant-gradient acceleration/deceleration.

For constant-gradient acceleration/deceleration, the gradient is determined by the rapid traverse rate and the time constant. The reaching time is determined by interpolation distance; therefore, the cycle time is shorter than acceleration/deceleration with fixed time constant.

In case of acceleration/deceleration with fixed time constant:



In case of constant-gradient acceleration/deceleration:



$T1 = Ts1 + Ts2$

$T2 = 2 \times \sqrt{Ts1 \times L/rapid}$

$Td = Ts2 + (0 \text{ to } 14 \text{ ms})$

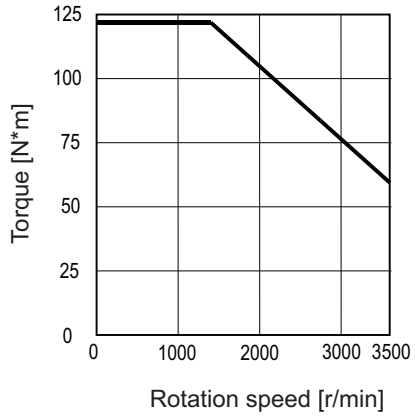
$\theta = \tan^{-1}\left(\frac{rapid}{Ts1}\right)$

rapid: Rapid traverse rate
Ts1: Acceleration/deceleration time
Ts2: Acceleration/deceleration time to reach the maximum speed
Td: Command deceleration check time
θ: Acceleration/deceleration gradient
T1: Interpolation time (Acceleration/deceleration with fixed time constant)
T2: Interpolation time (Constant-gradient acceleration/deceleration)
L: Interpolation distance

4.4.3 Rapid Traverse Constant-Gradient Multi-Step Acceleration/Deceleration

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	—	—	—	—	—	—	—

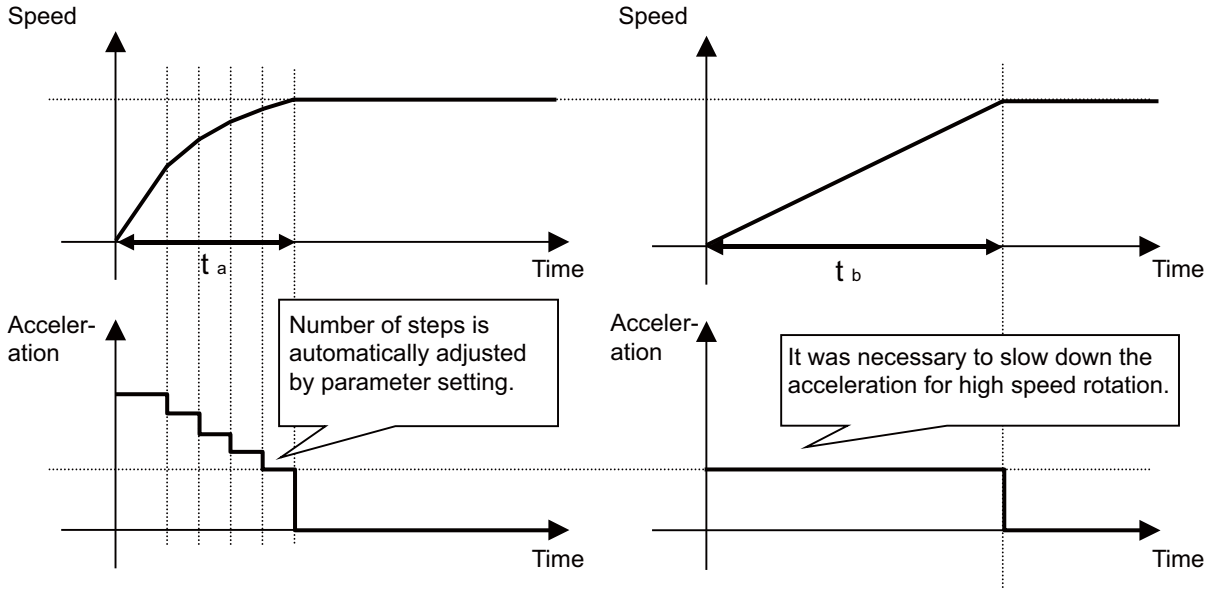
This function carries out the acceleration/deceleration according to the torque characteristic of the motor in the rapid traverse mode during automatic operation. (This function is not available in manual operation.) The rapid traverse constant-gradient multi-step acceleration/deceleration method makes for improved cycle time because the positioning time is shortened by using the motor ability to its maximum.
In general, the servo motor has the characteristic that the torque falls in the high-speed rotation range.



Note

(1) This characteristic is data at input voltage 380VAC.

In the rapid traverse constant-gradient acceleration/deceleration method, the acceleration has been treated constantly because this torque characteristic is not considered. So, it is necessary to use a minimum acceleration within the used speed range. Therefore, the margin of acceleration must be had in a low-speed range. Or if the acceleration is used to its maximum, the upper limit of the rotation speed must be slowed.
Then, to use the servo motor ability to its maximum, acceleration/deceleration to which the torque characteristic is considered is carried out by the rapid traverse constant-gradient multi-step acceleration/deceleration method.
The acceleration/deceleration patterns in the case where rapid traverse constant-gradient multi-step acceleration/deceleration are performed are as follows.



(a) Rapid traverse constant-gradient multi-step acceleration/deceleration

(b) Rapid traverse constant-gradient acceleration/deceleration

4.5 Thread Cutting

4.5.1 Thread Cutting (Lead/Thread Number Designation)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	△	△	△	△	○	○	○
L	○	○	○	○	○	○	○

Thread cutting with designated lead can be performed. Designate the number of threads per inch with the E address to cut inch threads.

■ Lead designation (Constant lead thread cutting)

The thread cutting with designated lead is performed based on the synchronization signals from the spindle encoder.

G33 Z/W__ X/U__ F/E__ Q__ L__ ;

G33	Thread cutting command
Z, W, X, U	Thread end point coordinates
F	Thread lead (normal lead threads)
E	Thread lead (precise lead threads)
Q	Shift angle at start of thread cutting (0.000° to 360.000°)
L	Lead axis No. (1 to maximum number of NC axes in a part system)

■ Thread number designation (Inch thread cutting)

Designate the number of threads per inch with the E address to cut inch threads.
Whether the E command is a thread number designation or lead designation is selected with the parameters.

G33 Z/W__ X/U__ E__ Q__ L__ ;

G33	Thread cutting command
Z, W, X, U	Thread end point coordinates
E	Thread number per inch
Q	Shift angle at start of thread cutting (0.000° to 360.000°)
L	Lead axis No. (1 to maximum number of NC axes in a part system)

[M system]

Thread cutting metric input

Input setting increment	B (0.001 mm)			C (0.0001 mm)		
Command address	F (mm/rev)	E (mm/rev)	E (ridges/inch)	F (mm/rev)	E (mm/rev)	E (ridges/inch)
Least command increment	1 (=0.001) (1.=1.000)	1 (=0.0001) (1.=1.0000)	1 (=1.00) (1.=1.00)	1 (=0.0001) (1.=1.0000)	1 (=0.00001) (1.=1.00000)	1 (=1.000) (1.=1.000)
Command range	0.001 to 999.999	0.0001 to 999.9999	0.03 to 999.99	0.0001 to 999.9999	0.00001 to 999.99999	0.026 to 222807.017

Input setting increment	D (0.00001 mm)			E (0.000001 mm)		
Command address	F (mm/rev)	E (mm/rev)	E (ridges/inch)	F (mm/rev)	E (mm/rev)	E (ridges/inch)
Least command increment	1 (=0.00001) (1.=1.00000)	1 (=0.000001) (1.=1.000000)	1 (=1.0000) (1.=1.0000)	1 (=0.000001) (1.=1.000000)	1 (=0.0000001) (1.=1.0000000)	1 (=1.00000) (1.=1.00000)
Command range	0.00001 to 999.99999	0.000001 to 999.999999	0.0255 to 224580.0000	0.000001 to 999.999999	0.0000001 to 999.9999999	0.02541 to 224719.00000

Thread cutting inch input

Input setting increment	B (0.0001 inch)			C (0.00001 inch)		
Command address	F (inch/rev)	E (inch/rev)	E (ridges/inch)	F (inch/rev)	E (inch/rev)	E (ridges/inch)
Least command increment	1 (=0.0001) (1.=1.0000)	1 (=0.00001) (1.=1.00000)	1 (=1.000) (1.=1.000)	1 (=0.00001) (1.=1.00000)	1 (=0.000001) (1.=1.000000)	1 (=1.0000) (1.=1.0000)
Command range	0.0001 to 39.3700	0.00001 to 39.37007	0.025 to 9999.999	0.00001 to 39.37007	0.000001 to 39.370078	0.0255 to 9999.9999

Input setting increment	D (0.000001 inch)			E (0.0000001 inch)		
Command address	F (inch/rev)	E (inch/rev)	E (ridges/inch)	F (inch/rev)	E (inch/rev)	E (ridges/inch)
Least command increment	1 (=0.000001) (1.=1.000000)	1 (=0.0000001) (1.=1.0000000)	1 (=1.00000) (1.=1.00000)	1 (=0.0000001) (1.=1.0000000)	1 (=0.00000001) (1.=1.00000000)	1 (=1.000000) (1.=1.000000)
Command range	0.000001 to 39.370078	0.0000001 to 39.3700787	0.02541 to 9999.99999	0.0000001 to 39.3700787	0.00000001 to 39.37007873	0.025401 to 9999.999999

Note

(1) It is not possible to assign a lead that causes the feedrate converted into the feed per minute value to exceed the maximum cutting feedrate.

[L system]

The thread lead by F command or E command (precision) is as follows depending on the parameter and whether the command value includes the decimal point.

Whether to handle the E command as the precision thread lead command or the number of inch threads depends on the MTB specifications.

F1 command and E1 command in the table are commands without decimal point.

Inch/metric input		Metric input (mm)		Inch input (inch)	
Input unit system		B (0.001)	C (0.0001)	B (0.0001)	C (0.00001)
F	Minimum	0.0001	0.00001	0.000001	0.0000001
	Maximum	999.9999	99.99999	99.999999	9.9999999
	F1 command (*1)	0.0001	0.00001	0.000001	0.0000001
		0.0001		0.000001	
		0.01		0.0001	
E Precision lead designation	Minimum	0.00001	0.000001	0.0000001	0.00000001
	Maximum	999.99999	99.999999	99.9999999	9.99999999
	E1 command (*1)	0.00001	0.000001	0.0000001	0.00000001
		0.0001		0.000001	
E Thread number designation	Minimum	0.03	0.255	0.0101	0.10001
	Maximum	999.99	9999.999	9999.9999	999.99999
	E1 command	1.00	1.000	1.0000	1.00000

(*1) Any value of the upper, middle or lower row on the table will be applied depending on the setting of the parameters that switch the F command unit.

Note

(1) It is not possible to assign a lead that causes the feedrate converted into the feed per minute value to exceed the maximum cutting feedrate.

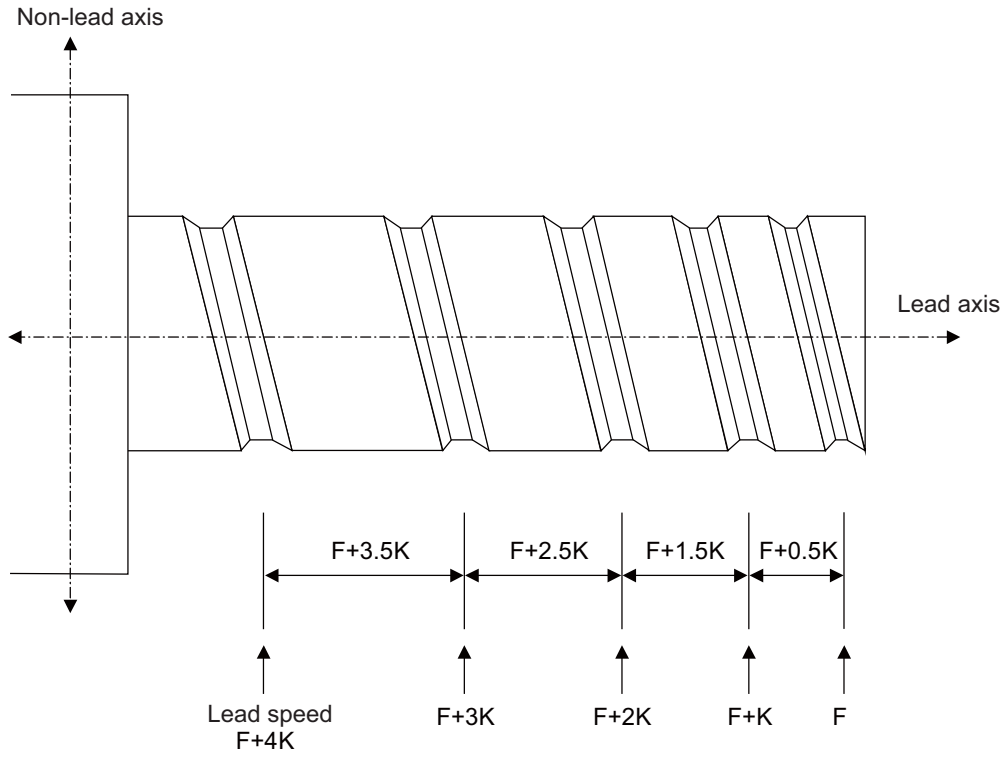
4.5.2 Variable Lead Thread Cutting

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	O	O	O	O	O	O	O

By commanding the lead increment/decrement amount per thread rotation, variable lead thread cutting can be done.
The machining program is commanded as follows.

G34 X/U__ Z/W__ F/E__ K__ ;

G34	Variable lead thread cutting command
X, U	Thread end point X coordinate
Z, W	Thread end point Z coordinate
F, E	Thread's basic lead
K	Lead increment/decrement amount per thread rotation



The lead is designated even when the the parameter for accurate thread cutting is disabled. (The number of screw threads is not designated.)

Inch/metric input	Metric input (mm)		Inch input (inch)	
Input unit system	B (0.001)	C (0.0001)	B (0.0001)	C (0.00001)
Command unit (*1)	0.00001	0.000001	0.0000001	0.00000001
	0.0001		0.000001	

(*1) Either value of the upper or lower row on the table will be applied depending on the setting of the parameters that switch the F command unit.

4.5.3 Synchronous Tapping

Note

(1) With digital I/F spindle

4.5.3.1 Synchronous Tapping Cycle

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

This function performs tapping through the synchronized control of the digital spindle and servo axis. This eliminates the need for floating taps and enables tapping to be conducted at a highly precise tap depth.

■ Tapping pitch assignment

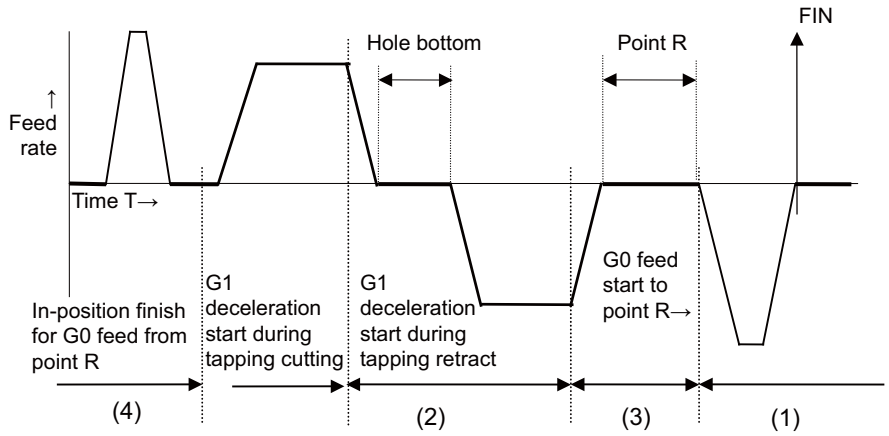
G84(G74) Xx1 Yy1 Zz1 Rr1 Dd1 Pp1 Ff1 Kk1 Ss1 ,Ss2 ,Rr2 ,Ii1 ,Jj1 Mm1 ;	
G84	: Mode, forward tapping
G74	: Mode, reverse tapping
Xx1,Yy1	: Hole position data, hole drilling position
Zz1	: Hole machining data, hole bottom position
Rr1	: Hole machining data, point R position
Dd1	: Tool spindle number (d is 1 to number of spindles) Depending on the parameter setting, command as "-d1" to carry out reverse tapping.
Pp1	: Hole machining data, dwell time at hole bottom
Ff1	: Z-axis feed amount (tapping pitch) per spindle rotation
Kk1	: Number of repetitions
Ss1	: Spindle rotation speed
,Ss2	: Spindle rotation speed during retract
,Rr2	: Synchronization method selection (r2=1 Synchronous, r2=0 Asynchronous)
,Ii1/,Jj1	: In-position width of positioning axis/hole drilling axis
Mm1	: M function designation

■ Tapping thread number assignment

G84(G74) Xx1 Yy1 Zz1 Rr1 Dd1 Pp1 Ee1 Kk1 Ss1 ,Ss2 ,Rr2 ,Ii1 ,Jj1 Mm1 ;	
G84	: Mode, forward tapping
G74	: Mode, reverse tapping
Xx1,Yy1	: Hole position data, hole drilling position
Zz1	: Hole machining data, hole bottom position
Rr1	: Hole machining data, point R position
Dd1	: Tool spindle number (d is 1 to number of spindles) Depending on the parameter setting, command as "-d1" to carry out reverse tapping.
Pp1	: Hole machining data, dwell time at hole bottom
Ee1	: Tap thread number per 1-inch feed of Z axis
Kk1	: Number of repetitions
Ss1	: Spindle rotation speed
,Ss2	: Spindle rotation speed during retract
,Rr2	: Synchronization method selection (r2=1 Synchronous, r2=0 Asynchronous)
,Ii1/,Jj1	: In-position width of positioning axis/hole drilling axis
Mm1	: M function designation

Note

- The synchronous tapping cycle can be used for axes other than the Z axis with the plane selection.
  Furthermore, in-position checks can be performed at the hole bottom or point R, etc. using the parameters.
  The figure below shows the correlation between the in-position width and the movement of the tapping axis of the synchronous tapping in-position check.



(1) Section where in-position check is performed using servo in-position width
(2) Section where in-position check is performed using in-position width for tapping
(3) Section where in-position check is performed using in-position width for cutting feed (G1, G2, G3)
(4) Section where in-position check is performed using in-position width for rapid traverse (G0)

4.5.3.2 Pecking Tapping Cycle

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

Designate the depth of cut per pass, and the depth to the hole bottom for a multiple number of passes. This reduces the load applied to the tool.
The amount retracted from the hole bottom is set to the parameters.
Select either the pecking tapping cycle or the deep-hole tapping cycle by parameter.
When the pecking tapping cycle is executed in the synchronous tapping mode, the synchronous tapping cycle function and pecking tapping cycle function are required.
When "depth of cutting per pass Q" is designated in the block containing the G84 or G74 command in the state where the pecking tapping cycle is selected by parameter, the pecking tapping cycle is executed.
Hole drilling can be performed by specifying the cutting reduction amount while the depth of cut in each cutting operation (step) is reduced.
In the following cases, the normal tapping cycle is established.

- When Q is not designated
- When the command value of Q is zero
- When there is no Pecking tapping cycle function

[M system]

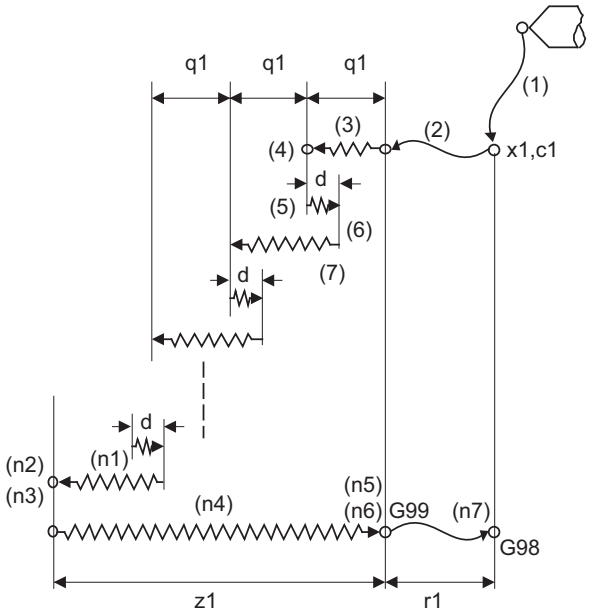
G84(G74) Xx1 Yy1 Zz1 Rr1 Qq1 Ff1(Ee1) Pp1 Ss1 ,Ss2 ,Ii1 ,Jj1 ,Rr2 Ll1 (Kk1) Jj2 ,Kk2 ;

G84	G84 forward tapping cycle
G74	G74 reverse tapping cycle
Xx1,Yy1	Hole drilling position
Zz1	Hole bottom position
Rr1	Point R position
Qq1	Depth of cutting in each pass (designated with incremental position)
Ff1	During synchronous tapping: Designation of drilling axis feed amount (tapping pitch) per spindle revolution (modal) During asynchronous tapping: Designation of the feedrate for cutting feed (modal)
Ee1	Tap thread number per 1-inch feed of Z axis
Pp1	Dwell time at hole bottom position
Ss1	Spindle rotation speed
,Ss2	Spindle rotation speed during retract
,Ii1	position width of positioning axis
,Jj1	position width of hole drilling axis
,Rr2	Synchronization method selection (r2=1:synchronous, r2=0:asynchronous)
Ll1	Number of repetitions
Kk1	Number of repetitions (It can be commanded by parameter.)
Jj2	Cutting reduction amount (When omitted, the mode will follow the setting of parameter.)
,Kk2	Minimum cutting amount (When omitted, the mode will follow the setting of parameter.)

[L system]

G84(G88, G84.1, G88.1) Xx1 Cc1 Zz1 Rr1 Qq1 Ff1(Ee1) Pp1 Ss1 ,Ss2 ,Ii1 ,Jj1 ,Rr2 Dd1 Kk1 Mm1 Jj2 ,Kk2 ;

G84	G84 Face forward tapping cycle
G88	G88 Side forward tapping cycle
G84.1	G84.1 Face reverse tapping cycle (It can be commanded by setting of parameter)
G88.1	G88.1 Side reverse tapping cycle (It can be commanded by setting of parameter)
Xx1,Cc1	Hole position data
Zz1	Hole bottom position
Rr1	Point R position
Qq1	Depth of cutting in each pass (designated with incremental position)
Ff1	During synchronous tapping: Designation of drilling axis feed amount (tapping pitch) per spindle revolution (modal) During asynchronous tapping: Designation of the feedrate for cuttingfeed (modal)
Ee1	Number of tap threads per 1-inch feed of Z axis
Pp1	Dwell time at hole bottom and point R return
Ss1	Spindle rotation speed
,Ss2	Spindle rotation speed during retract
,Ii1	In-position width of positioning axis
,Jj1	In-position width of hole drilling axis
,Rr2	Synchronization method selection (r2=1:synchronous, r2=0:asynchronous)
Dd1	Tapping spindle number assignment (By minus command, reverse tapping can be commanded by setting of parameter. It can be commanded when it is multiple-spindle control I. Program error occurs when multiple-spindle control II is commanded by D command.)
Kk1	Number of repetitions
Mm1	C axis clamp M code
Jj2	Cutting reduction amount (When omitted, the mode will follow the setting of parameter.)
,Kk2	Minimum cutting amount (When omitted, the mode will follow the setting of parameter.)



(1): G00 Xx1 Cc1 ,Ii1
(2): G00 Zr1
(3): G01 Zq1 Ff1
(4): M4 (Spindle reverse rotation)
(5): G01 Z-d Ff1
(6): M3 (Spindle forward rotation)
(7): G01 Z(q1+d) Ff1
:
(n1) G01 Z(z1-q1*n) Ff1
(n2) G04 Pp1
(n3) M4
(n4) G01 Z-z1 Ff1
(n5) G04 Pp1
(n6) M3
(n7) G00Z-r1 ,Ij1 G98 mode
No movement G99 mode

d : Retract amount (parameter)

Note

(1) This program is for the G84 command. The spindle forward rotation (M3) and reverse rotation (M4) are reversed with the G74 command.

4.5.3.3 Deep-Hole Tapping Cycle

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

In the deep-hole tapping, the load applied to the tool can be reduced by designating the depth of cut per pass and cutting the workpiece to the hole bottom for a multiple number of passes.
Adding this function, the pecking tapping cycle function is also added.
Under the deep-hole tapping cycle, the tool is retracted to the R-point every time.
Select either the pecking tapping cycle or the deep-hole tapping cycle by parameter.
When the deep-hole tapping cycle is executed in the synchronous tapping mode, the synchronous tapping cycle function and deep-hole tapping cycle function are required.
When "depth of cutting per pass Q" is designated in the block containing the G84 or G74 command in the state where the deep-hole tapping cycle is selected by parameter, the deep-hole tapping cycle is executed.
Hole drilling can be performed by specifying the cutting reduction amount while the depth of cut in each cutting operation (step) is reduced.
In the following cases, the normal tapping cycle is established.

- When Q is not designated
- When the command value of Q is zero
- When there is no deep tapping cycle function

[M system]

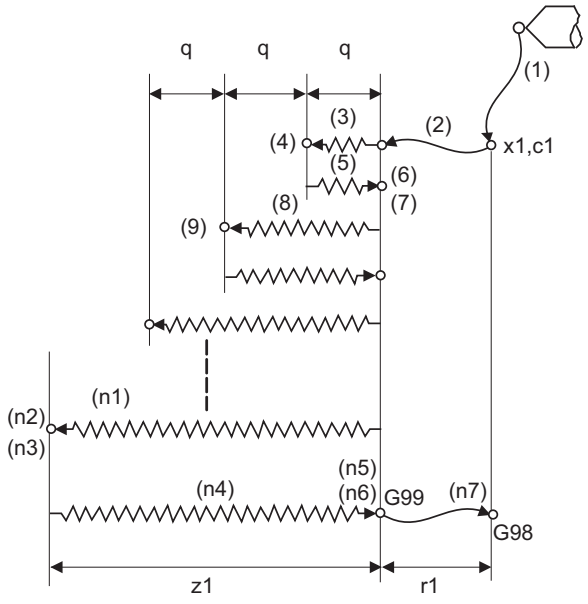
G84(G74) Xx1 Yy1 Zz1 Rr1 Qq1 Ff1(Ee1) Pp1 Ss1 ,Ss2 ,Ii1 ,Jj1 ,Rr2 Ll1 (Kk1) Jj2 ,Kk2 ;

G84	G84 forward tapping cycle
G74	G74 reverse tapping cycle
Xx1,Yy1	Hole drilling position
Zz1	Hole bottom position
Rr1	Point R position
Qq1	Depth of cutting in each pass (designated with incremental position)
Ff1	During synchronous tapping: Designation of drilling axis feed amount (tapping pitch) per spindle revolution (modal) During asynchronous tapping: Designation of the feedrate for cutting feed (modal)
Ee1	Tap thread number per 1-inch feed of Z axis
Pp1	Dwell time at hole bottom and point R return
Ss1	Spindle rotation speed
,Ss2	Spindle rotation speed during retract
,Ii1	In-position width of positioning axis
,Jj1	In-position width of hole drilling axis
,Rr2	Synchronization method selection (r2=1:synchronous, r2=0:asynchronous)
Ll1	Number of repetitions
Kk1	Number of repetitions (It commanded by parameter)
Jj2	Cutting reduction amount (When omitted, the mode will follow the setting of parameter.)
,Kk2	Minimum cutting amount (When omitted, the mode will follow the setting of parameter.)

[L system]

G84(G88, G84.1, G88.1) Xx1 Cc1 Zz1 Rr1 Qq1 Ff1(Ee1) Pp1 Ss1 ,Ss2 ,Ii1 ,Jj1 ,Rr2 Dd1 Kk1 Mm1 Jj2 ,Kk2 ;

G84	G84 Face forward tapping cycle
G88	G88 Side forward tapping cycle
G84.1	G84.1 Face reverse tapping cycle (It can be commanded by setting of parameter)
G88.1	G88.1 Side reverse tapping cycle (It can be commanded by setting of parameter)
Xx1,Cc1	Hole position data
Zz1	Hole bottom position
Rr1	Point R position
Qq1	Depth of cutting in each pass (designated with incremental position)
Ff1	During synchronous tapping: Designation of drilling axis feed amount (tapping pitch) per spindle revolution (modal) During asynchronous tapping: Designation of the feedrate for cutting feed (modal
Ee1	Number of tap threads per 1-inch feed of Z axis
Pp1	Dwell time at hole bottom and point R return
Ss1	Spindle rotation speed
,Ss2	Spindle rotation speed during retract
,Ii1	In-position width of positioning axis
,Jj1	In-position width of hole drilling axis
,Rr2	Synchronization method selection (r2=1:synchronous, r2=0:asynchronous)
Dd1	Tapping spindle number assignment (By minus command, reverse tapping can be commanded by setting of parameter. It can be commanded when it is multiple-spindle control I. Program error occurs when multiple-spindle control II is commanded by D command.)
Kk1	Number of repetitions
Mm1	C axis clamp M code
Jj2	Cutting reduction amount (When omitted, the mode will follow the setting of parameter.)
,Kk2	Minimum cutting amount (When omitted, the mode will follow the setting of parameter.)



(1):	G00 Xx1 Cc1	
(2):	G00 Zr1	
(3):	G01 Zq1 Ff1	
(4):	M4	(Spindle reverse rotation)
(5):	G01 Z-q1 Ff1	
(6):	G04 Pp1	
(7):	M3	(Spindle forward rotation)
(8):	G01 Z(2*q1)Ff1	
(9):		
:		
(n1)	G01 Zz1 Ff1	
(n2)	G4 Pp1	
(n3)	M4	
(n4)	G01 Z-z1 Ff1	
(n5)	G04 Pp1	
(n6)	M3	
(n7)	G00Z-r1	G98 mode
	No movement	G99 mode

Note

(1) This program is for the G84 command. The spindle forward rotation (M3) and reverse rotation (M4) are reversed with the G74 command.

4.5.3.5 Synchronous Tapping with Analog I/F Spindle

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

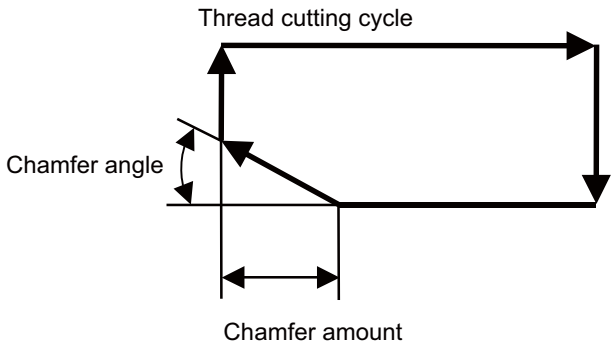
This function performs tapping using the analog-connected spindle. The functions in the table below can be used using the spindle analog-connected by inverter, etc. The operation of synchronous tapping cycle other than this function follows the specification in the table below.
When this function is used, the analog spindle with the function to control the spindle position needs to be connected. Also, the parameter needs to be turned ON.
The voltage value to be output to the analog spindle is between -10V and 10V, which is determined by the ratio of the spindle rotation speed corresponding to the command to analog spindle and the setting value of the parameter.

	Function name	G code
M system	Tapping cycle	G84
	Reverse tapping cycle	G74
L system	Face tapping cycle	G84
	Face reverse tapping cycle	G84.1
	Longitudinal tapping cycle	G88
	Longitudinal reverse tapping cycle	G88.1
	Synchronous tapping cycle	G84.2

4.5.4 Chamfering

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	O	O	O	O	O	O	O

Chamfering can be validated during the thread cutting cycle by using external signals.
The chamfer amount and angle are designated with parameters.



4.5.6 Circular Thread Cutting

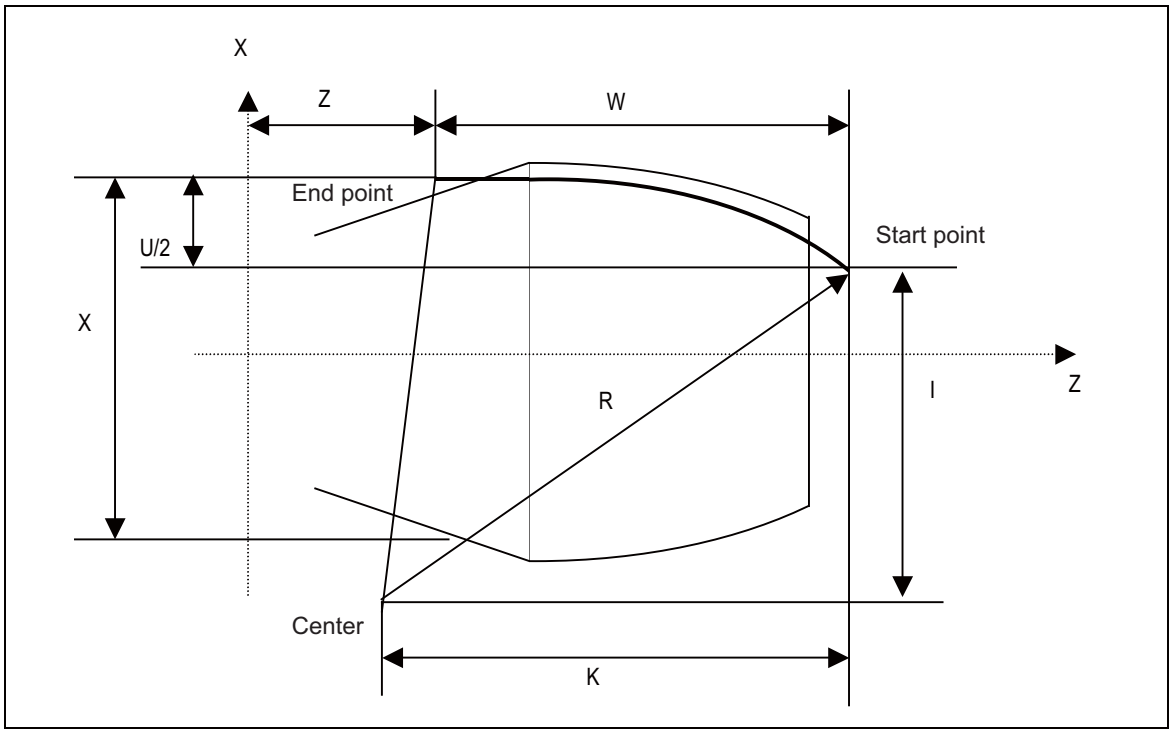
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	Δ	Δ	Δ	Δ	O	O	—

Circular thread in which the lead is in longitudinal direction can be cut.
This function can be used with the G code list 6 or 7.

■ Command format

G35(G36) Xx/Uu Zz/Ww { Ii Kk / Rr } Ff/Ee Qq ;

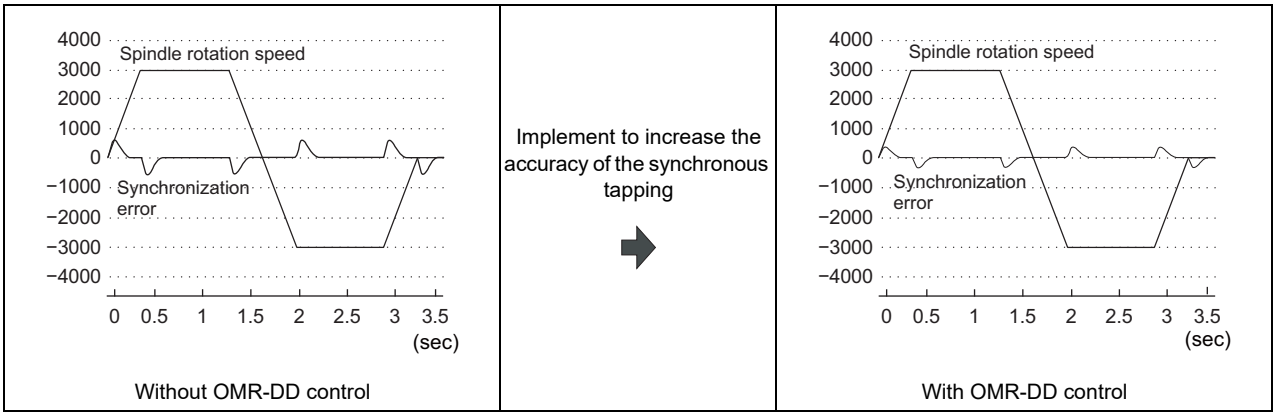
G35	Clockwise (CW)
(G36)	: Counterclockwise (CCW)
Xx/Uu	: X-axis arc end point coordinate
Zz/Ww	: Z-axis arc end point coordinate
Ii	: X-axis arc center (incremental value from start point to arc center)
Kk	: Z-axis arc center (incremental value from start point to arc center)
Rr	: Arc radius
Ff/Ee	: Longitudinal axis (axis with most travel) direction lead
	(Ff: normal lead thread cutting, Ee: precise lead threads or inch threads)
Qq	: Thread cutting start shift angle (0.000 to 360.000°)



4.5.8 High-Speed Synchronous Tapping (OMR-DD)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

The servo axis directly detects and compensates of the spindle's delay in tracking by using the communication between drive unit over the high-speed optical servo network. By minimizing the synchronization error, the accuracy of the synchronous tapping is increased.

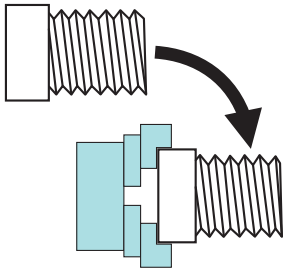


4.5.10 Thread Recutting

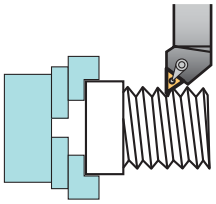
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	Δ	Δ	Δ	Δ	○	○	—

The function stores a thread groove position and compensates a start position of spindle thread cutting automatically so that the tool can pass along the memorized position of the thread groove at the thread cutting execution. Thread machined workpieces which have problems such as the thread number shortage, striped groove, etc., can be re-chucked and thread re-cut.

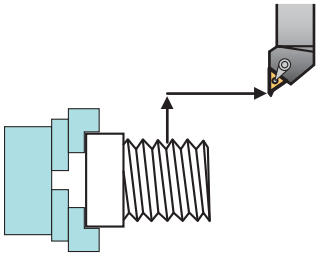
(1) Re-chucking of thread machined workpiece



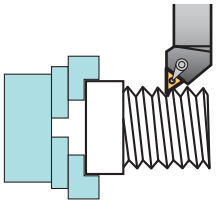
(2) Memorizing thread groove position (memorizing lead axis and spindle position)



(3) Moving to start position after tool retract



(4) Thread recutting ON and program operation start



The start position of spindle thread cutting is automatically compensated with the memorized position of the lead axis and spindle.

[Thread recutting correspondence table]

Thread cutting command	Z direction feed thread	X direction feed thread
	Right-handed/Left-handed thread	Scroll thread
Thread cutting	○	○
Continuous thread	Δ (*1)	Δ (*1)
Variable lead thread	Δ(*2)	Δ (*2)
Fixed Cycle for Turning Machining	○	○
Compound Type Fixed Cycle for Turning Machining	Δ(*3)	Δ(*3)

○: Machinable
Δ: Machinable with conditions
× : Impossible

Machinable with conditions (Δ)

(*1) Continuous thread: Conduct a memorizing of thread recutting position at the groove on the first block of continuous thread.
(*2) Variable lead thread: thread recutting is disable during an automatic operation with thread recutting enabled when the first thread cutting command is the variable lead thread cutting.
(*3) Compound type fixed cycle: Thread re-cutting can be performed only for the workpiece which has been cut its finishing allowance.

4.5.11 Thread Cutting Override

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	Δ	Δ	Δ	Δ	○	○	—

The thread cutting feedrate can be changed by changing the spindle override depending on rough cutting, finish machining, etc.
The spindle speed during thread cutting is determined with the spindle override at the start of thread cutting.

4.5.12 Variable Feed Thread Cutting

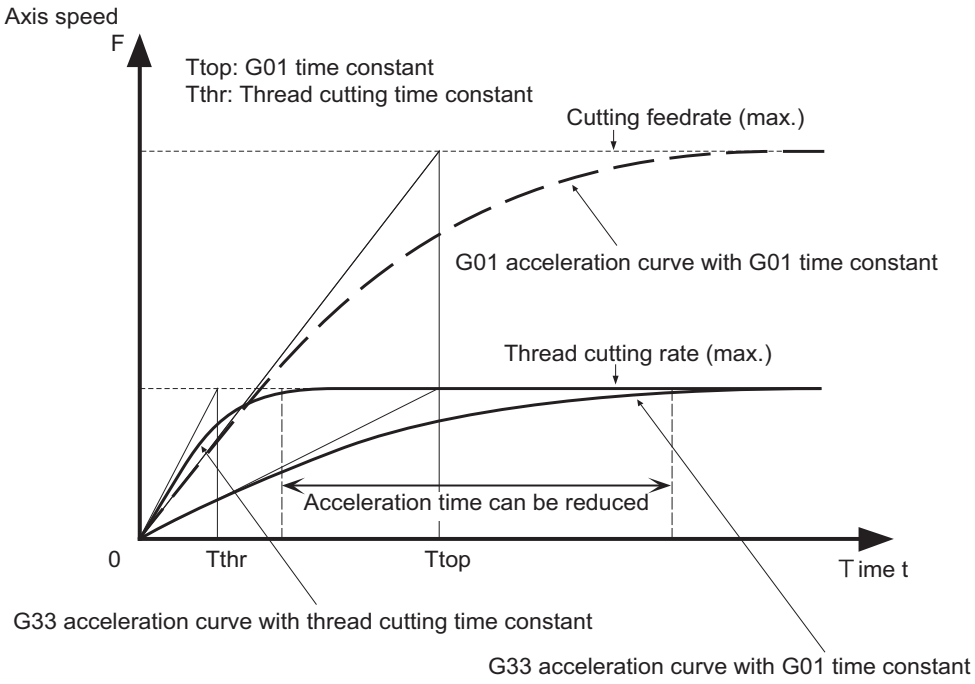
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	Δ	Δ	Δ	Δ	○	○	—

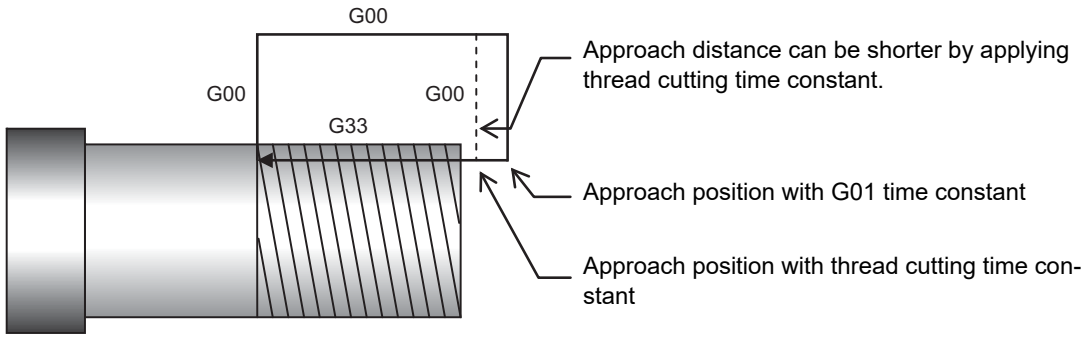
This function changes the cutting feedrate by the spindle override at the time of the thread cutting. The machining condition during thread cutting can be changed.

4.5.13 Thread Cutting Time Constant Switch

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

Thread cutting time constant can be applied to the acceleration/deceleration time constant of the NC control axis during the thread cutting. Usually, incorrect lead parts occur at the start and the end of thread cutting due to the acceleration/deceleration of the NC control axis. However, the acceleration/deceleration time, which causes incorrect lead parts, can be reduced by applying the thread cutting time constant to the NC control axis during the thread cutting so that the incorrect thread parts can be reduced. Additionally, the machining time can be reduced as much as the acceleration/deceleration time reduction of thread cutting.



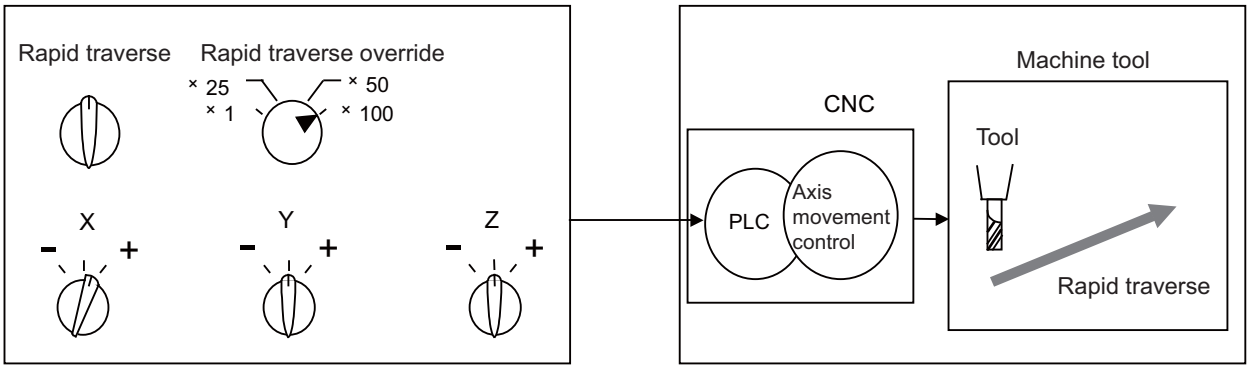


4.6 Manual Feed

4.6.1 Manual Rapid Traverse

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

When the manual rapid traverse mode is selected, the tool can be moved at the rapid traverse rate for each axis separately.
Override can also be applied to the rapid traverse rate by means of the rapid traverse override function.
Rapid traverse override can be set for each part system respectively.

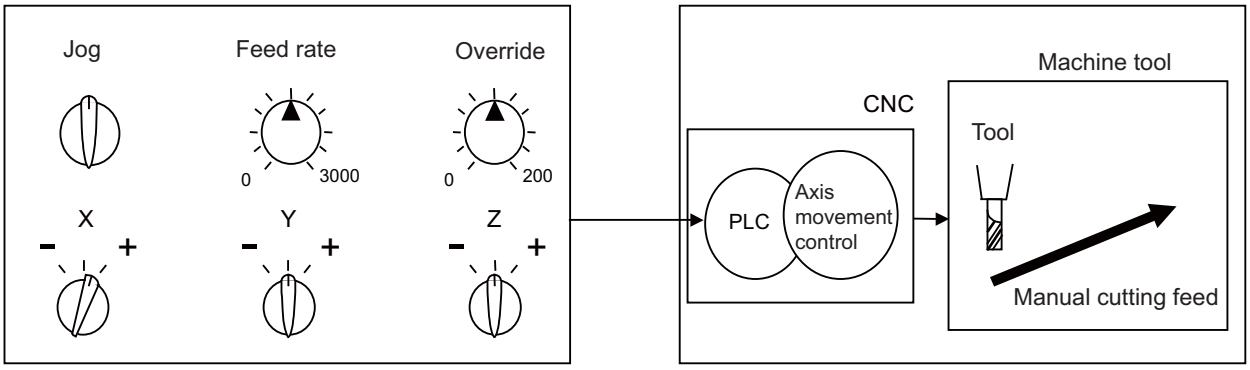


4.6.2 Jog Feed

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

When the jog feed mode is selected, the tool can be moved in the axis direction (+ or -) in which the machine is to be moved at the per-minute feed.
The jog feedrate can be set for each axis with the parameters.
If the jog feedrate is not set with the parameters, the jog feedrate is set with the PLC signal.

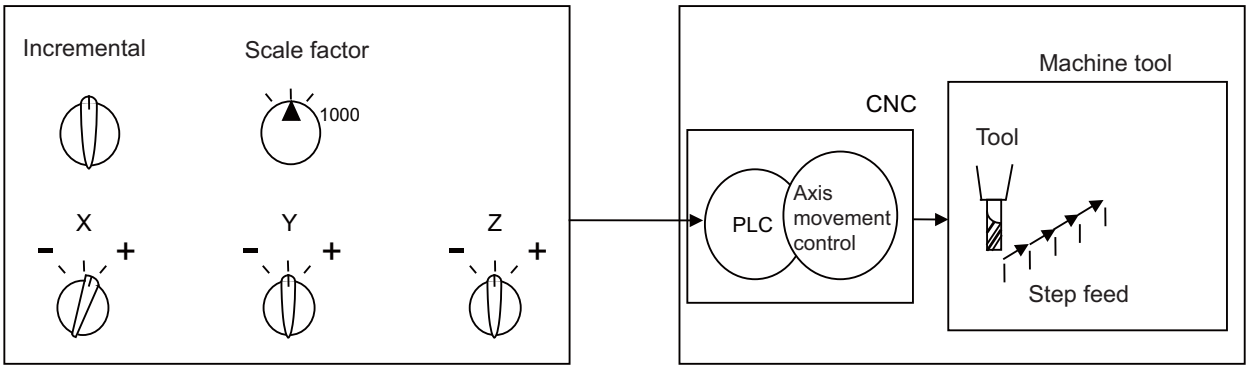
There are two methods for PLC signal which sets the jog feed rate: the code method and the value setting method.
The method to be used is selected by PLC signal for each part system. The signals of code and value setting methods are set for each part system.



4.6.3 Incremental Feed

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

When the incremental feed mode is selected, the tool can be operated by an amount equivalent to the designated amount (incremental amount) in the axis direction each time the jog switch is pressed. The incremental feed amount is the amount obtained by multiplying the least command increment that was set with the parameter by the incremental feed magnification rate.
The incremental feed amount parameter and its magnification rate can be set for each part system respectively.



4.6.4 Handle Feed

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

In the handle feed mode, the machine can be moved in very small amounts by rotating the manual pulse generator.
The scale can be selected from ×1, ×10, ×100, ×1000 or arbitrary value.
If the least command increment is 10 nm or 1 nm, the scale can be selected from ×5000, ×10000, ×50000 or ×100000, as well.
Individual axes can be moved in very small amounts separately by rotating the manual pulse generators installed on each of the axes.

Note

(1) The actual movement amount and scale may not match if the manual pulse generator is rotated quickly.

Up to three handles (manual pulse generators) can be used with tthis CNC.
The handles can be connected to the control unit and remote I/O unit. If the operation panel I/O unit is connected, the handle can also be connected to the operation panel I/O unit.

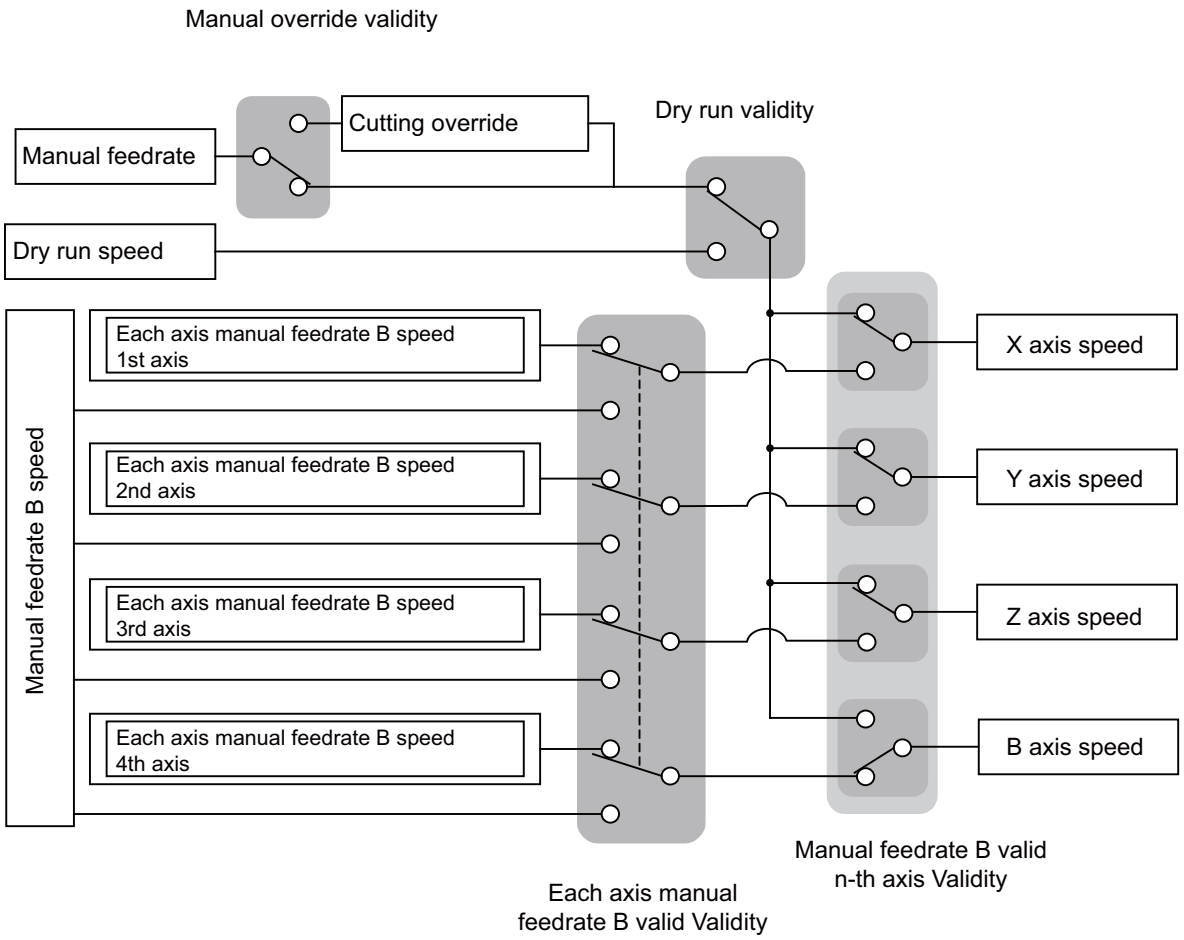
4.6.5 Manual Feedrate B

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

Manual feedrate B is a function that sets an arbitrary axis feedrate from the user PLC separately from the "manual feedrate". The "manual feedrate B" feedrate setting can be selected from the feedrate common for all axes and the feedrate independent of reach axis.
By combining the "manual feedrate B" function with the manual/automatic simultaneous function, an arbitrary axis can be moved at the "manual feedrate B" independently of the machining program operation even during automatic operation. Similarly, if the jog mode and other manual operation mode are set simultaneously, an arbitrary axis can be moved at a speed independent from the "manual feedrate" even during the manual operation mode.

The "manual feedrate B" function can move an axis at a speed different from the "manual feedrate". This is not affected by dry run, or by manual or cutting override, so an arbitrary axis can be moved independently even in operations during automatic operation or override during manual axis movement.

The relation of the "manual feedrate B" and "manual feedrate" is shown below.



Note

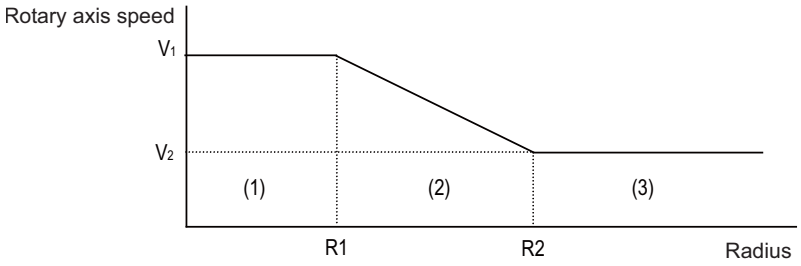
(1) For the axis to which manual feedrate B is applied, the feedrate is not displayed on the screen.

4.6.6 Manual Feedrate B Surface Speed Control

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	—	—	—
L	—	—	—	—	—	—	—

When using the manual feedrate B function and machining by moving the orthogonal axis while rotating the rotary table, the tool nose and workpiece's relative speed will drop as the tool nears the rotation center if the table rotation speed remains under the set conditions. This function controls the table rotation speed according to the distance from the rotation center.

As shown below, the distance (radius) from the rotation center at two points and the rotary axis speed at each point is set in the parameters. When the "Manual feedrate B surface speed control valid" signal is turned ON, the rotary axis speed will be automatically calculated according to the current radius R.



(1) If R ≤ R1, then V1 will be applied.
(2) If R1 < R < R2, the speed V is calculated with the following expression.

$$V=\frac{(V_2 - V_1)}{(R_2 - R_1)} *(R - R_1)+V_1$$

(3) If R2 ≤ R, then V2 will be applied.

Override can be applied in the range of 0 to 200% in respect to the rotary axis's speed for which the manual feedrate B surface speed control is valid.

This function can be used with a rotary axis for which the manual feedrate B function is valid.
The manual feedrate B speed and each axis' manual feedrate B speed which are issued from the user PLC is ignored for an axis for which this function is valid.

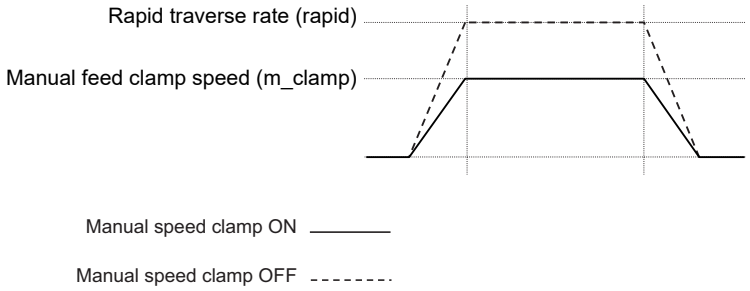
4.6.8 Manual Speed Clamp

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

The maximum speed for manual feed can be switched to the rapid traverse rate or the manual feed clamp speed.
This function is usable, for instance, when the axis speed needs to be clamped at an arbitrary rate for performing a manual feed with a cutting cover opened.

The clamp speed is switched with the parameter or PLC signal.
The manual feed clamp speed can be set to an arbitrary speed with the parameter.
The manual feedrate can be applied for operation modes as follows:

- Jog mode
- Handle mode
- Incremental mode
- Manual reference position return (high-speed) mode



4.7 Dwell

With this function, the program command temporarily stops the machine movement and puts the machine into standby status. Allowing the starting time of the next block to be delayed. The waiting time state can be canceled by inputting the skip signal.

4.7.1 Dwell (Time-Based Designation)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

When G04 is commanded in the asynchronous feed mode (G94), the machine waits for the specified amount of time before executing the next block.

■ Command format

G94 G04 X/U__ ; or G94 G04 P__ ;
X/P/U : Dwell time

- G94 is not necessary in the asynchronous feed mode (G94).
- The decimal point command is valid for dwell time designated with the address "X" or "U".
- The decimal point command is valid for dwell time designated with the address "P", depending on the parameter setting. When the decimal point command is set to be invalid by the parameter, the number below the decimal point of the P command value is ignored.
- When the decimal point command is valid or invalid, the dwell time command range for each status is as follows.

Command range when the decimal point command is valid	Command range when the decimal point command is invalid
0 to 99999.999 (s)	0 to 99999999 (ms)

- When the dwell is commanded with no decimal point, the setting unit for dwell time can be changed by setting the parameter.

4.7.2 Dwell (Revolution-Based Designation)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

When G04 is commanded in the synchronous feed mode (G95), the machine waits for the spindle to rotate for the number of the revolutions designated.

■ Command format

G95 G04 X/U__ D__ ; or G95 G04 P__ D__ ;	
X/P/U	: Number of revolutions for dwell
D	: Dwell target spindle (For command using the spindle number, the spindle number should be within the number of spindles for specifications. For command using the spindle name, the number from 1 to 9 is valid for the spindle name.)

- When there is a D command in the same block as G04, dwell (revolution-based designation) is executed regardless of the parameter setting and irrespective of asynchronous feed mode (G94) or synchronous feed mode (G95).
- When there is no D command, the setting will be as follows depending on the parameter setting to enable dwell (revolution-based designation).

Parameter setting	
Valid	Dwell (revolution-based designation) is executed for the first spindle.
Invalid	Dwell (time-based designation) is executed.

- G95 is not necessary in the synchronous feed mode (G95).
- The decimal point command is valid for the number of revolutions for dwell designated with the address "X" or "U".
- The decimal point command is valid for the number of revolutions for dwell designated with the address "P", depending on the parameter setting. When the decimal point command is set to be invalid by the parameter, the number below the decimal point of the P command value is ignored.
- When the decimal point command is valid or invalid, the command range of number of revolution for dwell is as follows for each status.

Command range when the decimal point command is valid	Command range when the decimal point command is invalid
0 to 99999.999 (rev)	0 to 99999999 (0.001 rev)

- When the dwell is commanded with no decimal point, the setting unit for the number of revolution can be changed by setting the parameter.

5

Program Memory/Editing

5.1 Memory Capacity

Machining programs are stored in the NC memory, NC memory 2, DS, and external memory device (front SD card, built-in disk of display unit, etc.).
When using devices such as the built-in disk (HD) of the display unit, DS and memory card, mass-editing, which is carried out on those devices, is possible.
The data size that can be handled in the mass-editing differs depending on the devices.

5.1.1 Memory Capacity (Number of Programs Stored)

[M system]

Memory capacity (number of programs stored)	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
500 KB [1280 m] (1000 programs)	○	○	○	○	○	○	○
1000 KB [2560 m] (1000 programs)	Δ	Δ	Δ	Δ	—	—	—
2000 KB [5120 m] (1000 programs)	Δ	Δ	Δ	Δ	—	—	—

[L system]

Memory capacity (number of programs stored)	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
500 KB [1280 m] (1000 programs)	○	○	○	○	○	○	○
1000 KB [2560 m] (1000 programs)	Δ	Δ	Δ	Δ	—	—	—
2000 KB [5120 m] (1000 programs)	Δ	Δ	Δ	Δ	—	—	—

Note

(1) The tape length for the multi-part system specifications is the total for all part systems.

5.1.2 Extended Memory

[M system]]

Memory capacity (number of programs stored)	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
2000 KB [5120 m] (1000 programs)	○	○	○	○	○	○	○

[L system]]

Memory capacity (number of programs stored)	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
2000 KB [5120 m] (1000 programs)	○	○	○	○	○	○	○

To use the extended memory, select the device "NC memory 2".

Note

(1) The tape length for the multi-part system specifications is the total for all part systems.

5.2 External Signal Display Switch

5.2.1 Program Editing

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

The following editing functions are possible.

■ Program erasing

- Machining programs can be erased individually or totally.

■ Program filing

- This function displays a list of the machining programs stored (registered) in the controller memory.
- The programs are displayed in ascending order.
- Comments can be added to corresponding program numbers.

■ Program copying

- Machining programs stored in the controller memory can be copied, condensed or merged.
- The program No. of the machining programs in the memory can be changed.

■ Program editing

- Overwriting, inserting and erasing can be done per character.

■ Mass editing

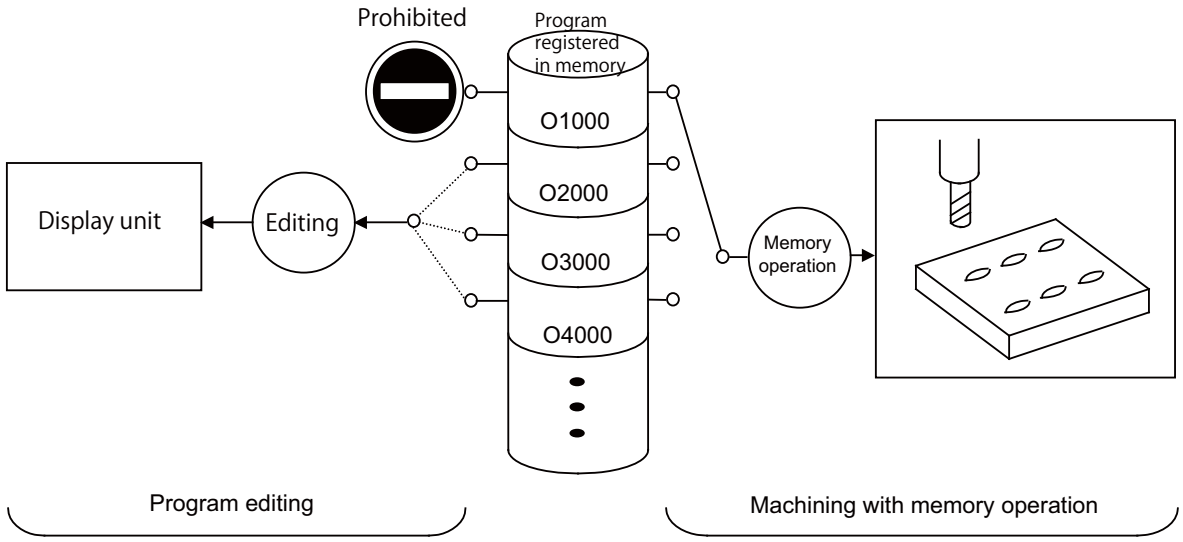
There are regular editing and mass-editing for program editing. The specification and restrictions are different between the regular editing and mass-editing.
Refer to the table below for conditions to be mass-editing and its maximum editing size.

Model	Conditions to be mass-editing	Maximum editing size
M800V Series M80VW Series	• The storage location of the machining program to be edited is either a built-in disk (HD) of the display unit, a USB memory, a memory card or a DS. • A file size is 1.0 MB or larger. (The size could be 2.0 MB or larger, depending on the parameter settings.)	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. • A file size is 0.5 MB or larger.	

5.2.2 Background Editing

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

This function enables one machining program to be created or editing while another program is being run.

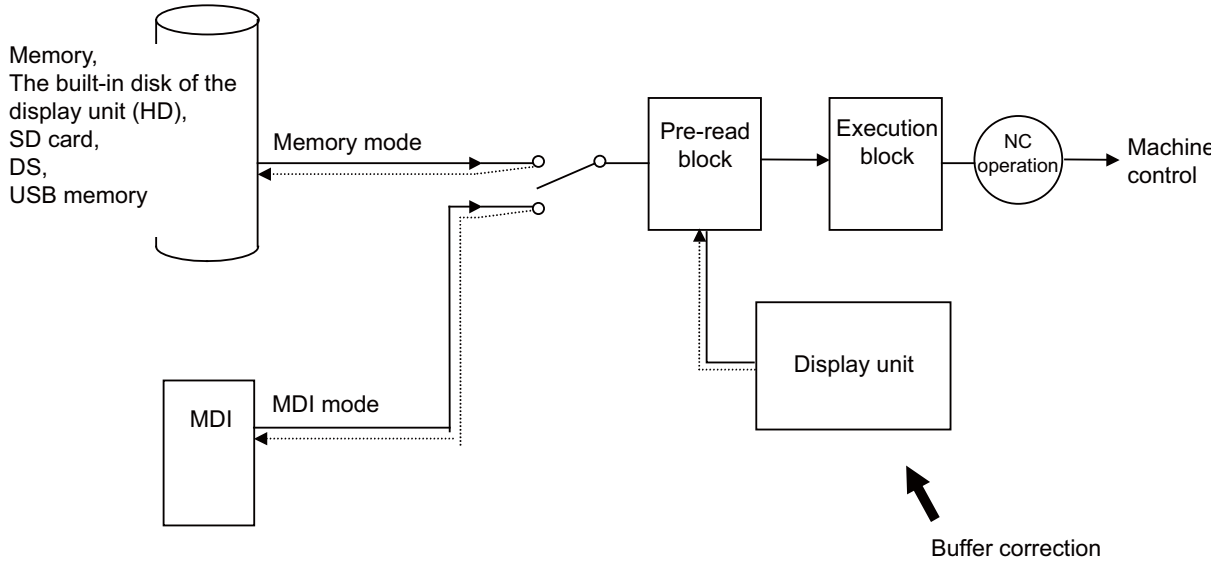


- The data of the machining programs being used in memory operation can be displayed and scrolled on the setting and display unit, but data cannot be added, revised or deleted.
- The editing functions mentioned in the preceding section can be used at any time for machining programs which are not being used for memory operation.
  This makes it possible to prepare and edit the next program for machining, and so the machining preparations can be made more efficiently.
- The machining program will not be searched as the operation target even when searched in the edit screen.

5.2.3 Buffer Correction

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

During automatic operation (NC memory, NC memory 2, SD card, USB memory or DS operation) or MDI operation, this function initiates single block stop and enables the next command to be corrected or changed.
Only memory or DS operation allows the changes with buffer corrections to be updated in the machining program.
When a program error has occurred, the function enables the block in which the error occurred to be corrected and operation to be resumed without having to perform NC resetting.



5.2.5 Multi-Part System Simultaneous Program Editing

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	—	—	—
L	○	○	○	○	○	○	○

When an operation to open a machining program in the NC memory is performed on the edit screen, machining programs are opened in the right and left areas at the same time; the specified machining program of the displayed part system in the edit area being selected and the machining program of another part system with the same name in the unselected edit area.

5.2.6 Special Program Editing Display for Synchronization Between Part Systems

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

Pressing the [Synchro view] menu enables synchronized display of the left- and right-side programs aligned using the timing synchronization symbols, when all the following conditions are met:

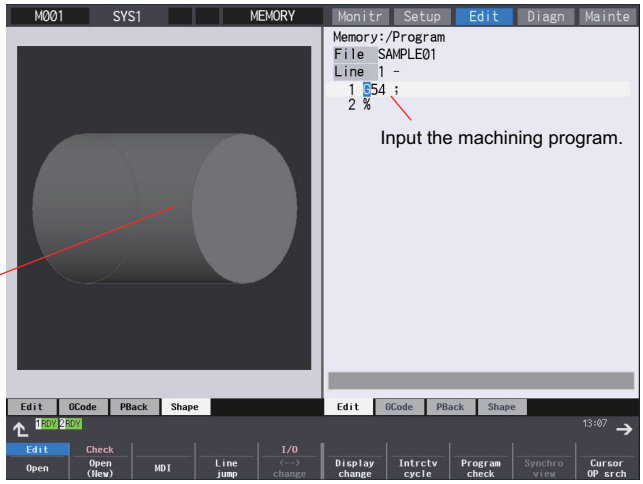
- Multi-part system program management is ON
- Multi-program display type is selected
- The left and right edit areas are displaying the same named programs of different part systems stored on the NC memory

5.2.7 Finish Shape View Programming

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

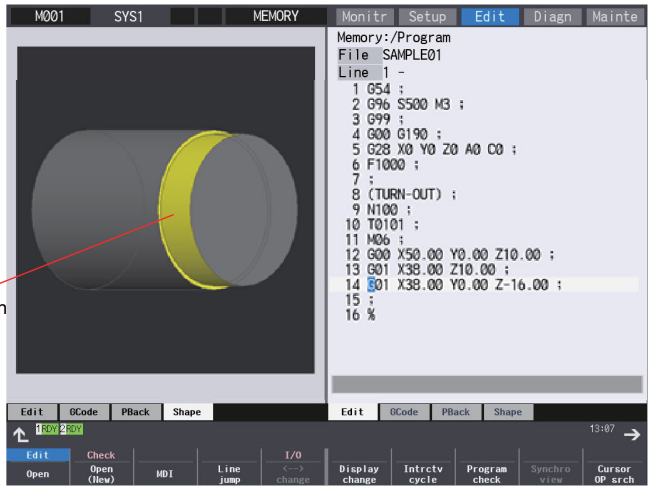
This function is used to display the machining shape according to the commands of the machining program input on the edit screen, and is intended to eliminate mistakes in programming. Therefore, the machining shape can be confirmed easily without performing the automatic operation or the graphic check.

(Example) For a new program file, confirm the finish shape by inputting the machining program.



The input machining program will be reflected in the shape.
* In this example, as the cutting is not performed with G54, the shape will not be changed.

Each time a block of machining program is input, the machining shape is drawn graphically.



The input machining program will be reflected in the shape.

6

Operation and Display

6.1 Structure of Operation/Display Panel

Setting display unit consists of the display and keyboard unit.
Refer to the "Specifications Manual (Hardware)" for details.

Note

(1) For Separated-type color touchscreen display (19-type LCD TFT/Windows10), only software keyboard is available and there is no hardware keyboard.

6.1.1 Color Display (8.4-Type LCD TFT)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	□	□
L	—	—	—	—	—	□	□

6.1.2 Color Touchscreen Display (10.4-Type LCD TFT)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	□	□	—	□	□
L	—	—	□	□	—	□	□

6.1.3 Color Touchscreen Display (15-Type LCD TFT)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	□	□	—	□	□
L	—	—	□	□	—	□	□

6.1.4 Color Touchscreen Display (19-Type LCD TFT)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	□	□	—	□	□
L	—	—	□	□	—	□	□

6.1.8 Separated-Type Color Touchscreen Display (15-Type LCD TFT/Windows10)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	—	—	□	—	—
L	□	□	—	—	□	—	—

6.1.9 Separated-Type Color Touchscreen Display (19-Type LCD TFT/Windows10)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	—	—	□	—	—
L	□	□	—	—	□	—	—

6.1.10 Separated-Type Color Touchscreen Display (19-Type Horizontal LCD TFT/ Windows10)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	—	—	□	—	—
L	□	□	—	—	□	—	—

6.2 Operation Methods and Functions

6.2.1 Operation Input

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

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

Numeric values, function symbols, operators and parentheses ( ) are combined and set in the data setting area.
Press the INPUT key to display the operation results. Press the INPUT key again to process the data and display the result on the screen. The contents in the data setting area are erased.

Examples of operator settings and results			Function symbols, setting examples and results			
Operation	Setting example	Operation results	Function	Function symbol	Setting example	Operation results
Addition	= 100+50	150.000	Absolute value	ABS	= ABS(50-60)	10
Subtraction	= 100-50	50.000	Square root	SQRT	= SQRT(3)	1.732
Multiplication	= 12.3*4	49.200	Sine	SIN	= SIN(30)	0.5
Division	= 100/3	33.333	Cosine	COS	= COS(15)	0.966
Function	= 1.2*(2.5+SQRT(4))	5.4	Tangent	TAN	= TAN(45)	1
			Arc tangent	ATAN	= ATAN(1.3)	52.431

6.2.2 Absolute/Incremental Setting

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

When setting the data, the absolute/incremental setting can be selected from the menu.

The absolute/incremental settings can be selected on the following screens.
- Common variable screen
- Tool compensation amount screen
- Coordinate system offset screen

6.2.5 Displayed Part System Switch

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	—
L	○	○	○	○	○	○	○

The part system displayed on the screen can be changed with the [$<->$] keys.
The number of displayed part systems is counted by one each time the [$<->$] keys are pressed. The screen corresponding to that part system opens.
If the number of displayed part systems exceeds the valid number of part systems, the number of displayed part systems will return to 1.

6.2.6 Menu List

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

The menu list function displays the menu configuration of each screen as a list making it possible to directly select the menu for other screens.
When the cursor is moved to the menu, the outline of that menu's functions will also appear. The menu can be selected while checking the details of the menu.

6.2.7 Display Switch by Operation Mode

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

The screen display changes when the screen mode selection switch is changed.
The details corresponding to the operation mode are displayed.

6.2.8 External Signal Display Switch

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	—
L	○	○	○	○	○	○	—

The screen display changes with the signal from PLC.

6.2.9 Screen Saver

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

The screen saver function protects the screen display unit by turning the backlight OFF after the time set in the parameters has elapsed. The backlight is turned OFF after a certain period of time (automatic change function) or after the key operations (manual change function).
The screen is displayed again by pressing any key, or by touching anywhere on the screen if the display unit carries a touch-sensitive screen.

6.2.10 Parameter Guidance

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

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

If the [?] key is pressed on any screen, the parameter guidance window will open. If a pop-up window other than the parameter guidance window is opened, the parameter guidance window will open over the currently opened pop-up window.
For the display unit with a touch-sensitive screen, if you hold down the parameter data area (tap the target and keep your finger down for a while), the guidance of that parameter is displayed.

6.2.11 Alarm Guidance

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

The alarm guidance displays the alarm message, details and remedy for the alarm currently issued.
By utilizing the guidance information, identify the cause from possible factors and determine the countermeasures.
When a multiple number of alarms are issued at the same time, guidance will be displayed for all the alarm issued.

6.2.12 Machining Program Input Mistake Check Warning

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	—	—	—
L	Δ	Δ	Δ	Δ	—	—	—

If an illegal input is found in the decimal point after the current cursor position, the cursor will move to that position, and a warning message will appear.
If this function is ON when editing the program, the decimal point will be checked for the block each time an edit key (alphabet, number, symbol, Delete, etc.) is pressed. The block is not checked when the cursor keys or page feed keys are pressed.
The warning for illegal machining program can also be issued while editing an MDI program.
The comment block is also subject to the warning for illegal machining program.

A warning does not appear in the following cases.
- When the data in the address subject to the illegal decimal point input is "0", a warning will not be issued regardless of whether there is a decimal point or not. (Example: A warning is not issued for "X0".)
- When the data in the address subject to the illegal decimal point input is omitted, a warning will not be issued. (Example: A warning is not issued for "G28XYZ".)
- Blocks containing "[" or "]" are not subject to the warning for illegal machining program.

6.2.14 Screenshot Capture

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	○	○	—	○	○
L	—	—	○	○	—	○	○

This function allows to output a bitmap file of a screen displayed on the setting and display unit to USB memory or SD card.

6.2.15 User Selectable Menu Configuration

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

This function allows to change the display order of operations, procedure and edit screen, and to change display/non-display selection.

6.2.16 PC-NC Network Automatic Connection

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	—	—	○	—	—
L	○	○	—	—	○	—	—

This function supports to restore the connection when the network connection cannot be created between the display unit and the control unit.
When the connection is not established even after the time out has expired, the connectable control unit IP address list appears. From the IP address list, the network connection can be re-established and restored when the IP address, which is to be connected with the control unit, is selected.

6.2.17 Device Open Parameter

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

This function can set or change the user backed up area of the PLC device from the NC screen.
The following settings are available from the NC screen.
- Divide the device area accordance with the specifications of the machine maker and set and display for each divided area.
- Switch the display format or data type for each divided area.

6.2.18 SRAM Open Parameter

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

This function can set or change the SRAM open area for the machine maker from the NC screen.
The following settings are available from the NC screen.
- Divide the SRAM area accordance with the specifications of the machine maker and set and display for each divided area.
- Switch the display format or data type for each divided area.

6.2.19 MTB Selectable Menu Configuration

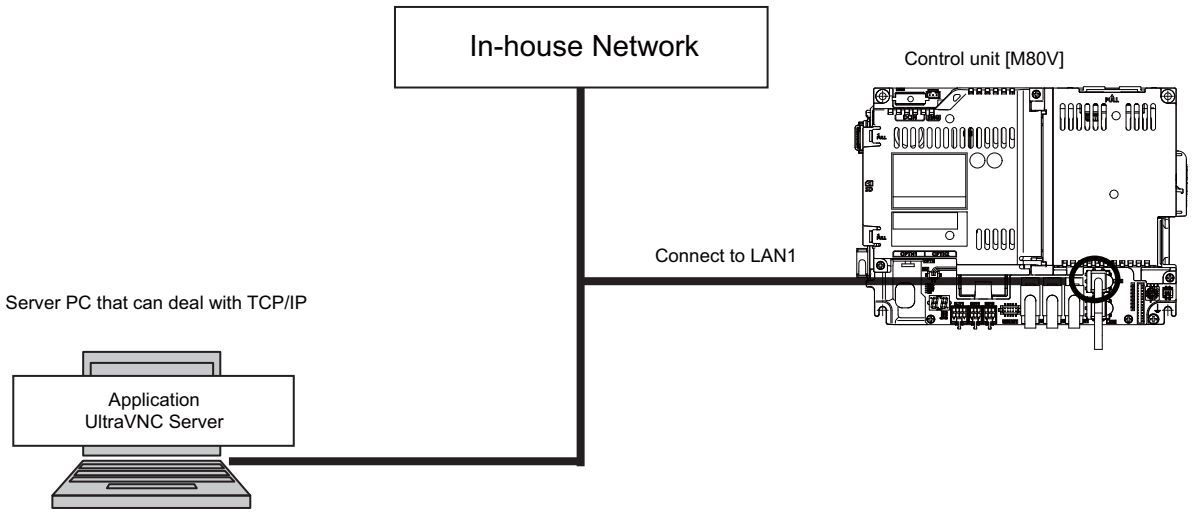
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

Menu items on the "Monitor", "Setup", and "Edit" screens of Mitsubishi Electric standard format can be moved within a screen or hidden as desired.
The custom screen menu items added by machine tool builders, to the contrary, cannot be moved or hidden.

6.2.20 Remote Desktop Connection

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	Δ	Δ	—	○	○
L	—	—	Δ	Δ	—	○	○

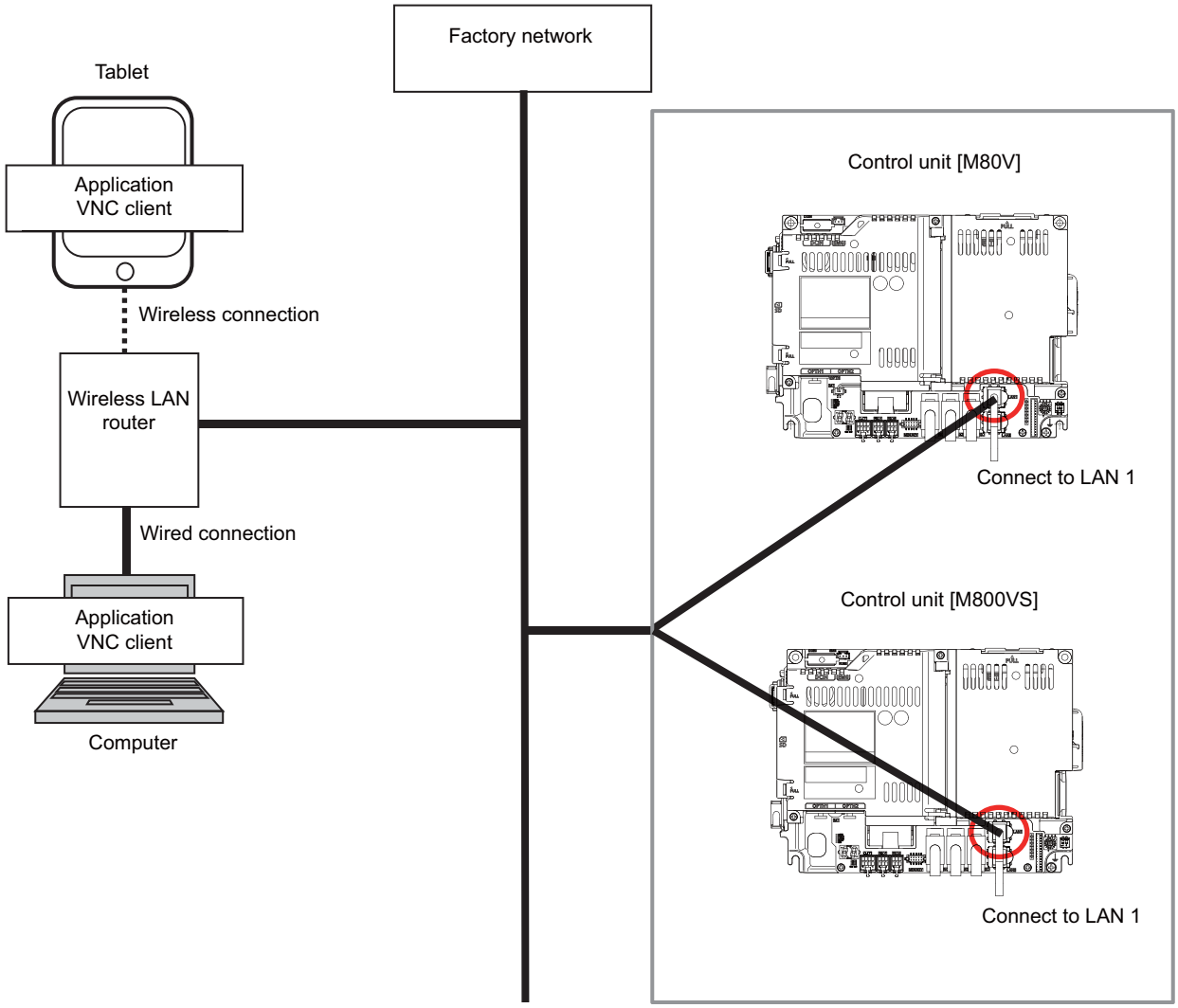
Remote desktop connection enables you to operate an external personal computer through the NC screen whenUltraVNC server is installed in the PC.



6.2.21 VNC Server

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	Δ	Δ	—	O	O
L	—	—	Δ	Δ	—	O	O

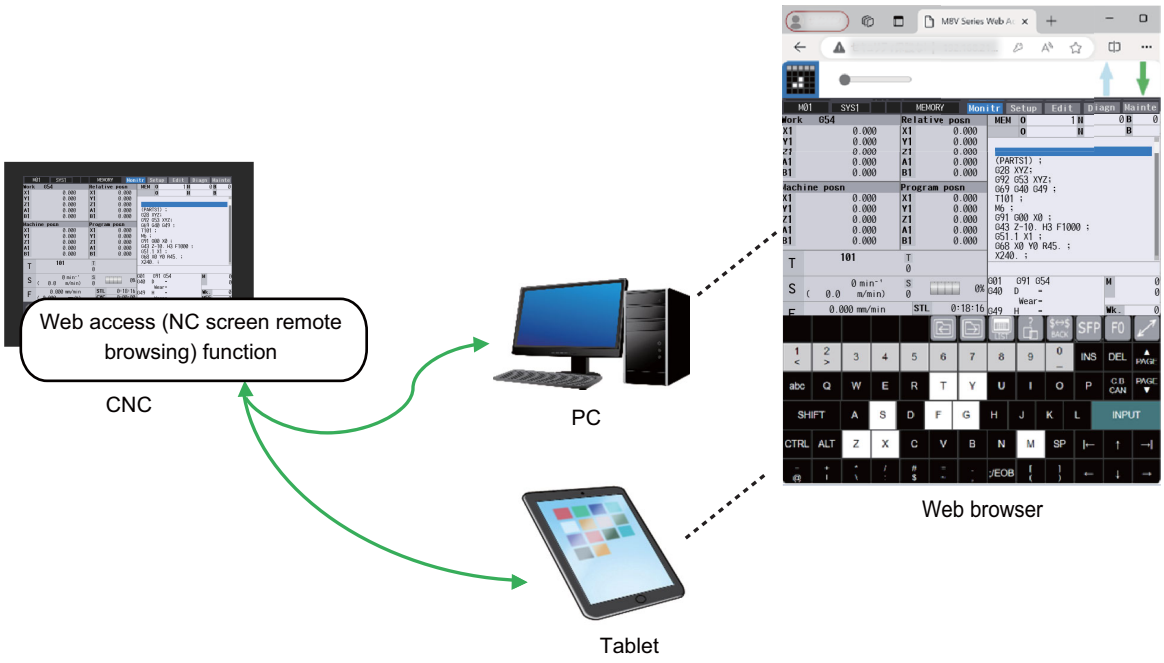
This function enables status confirmation and remote operation by displaying the NC screen display on the external computer or tablet PC.
This function can be valid by setting the parameter.



6.2.22 Web Access (NC Screen Remote Browsing)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	○	○	—	○	○
L	—	—	○	○	—	○	○

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. It can be used both wired and wireless.



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

6.3 Display Methods and Contents

6.3.1 Status Display

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

The status of the program now being executed is indicated.

- Display of G, S, T, M commands and 2nd miscellaneous command modal values
- Feed rate display
- Tool compensation No. and compensation amount display
- Real speed display(*1)

(*1) The feed rate of each axis is converted from the final speed output to the drive unit, and is displayed. However, during follow up, the speed is converted and displayed with the signals from the detector installed on the servo motor.

6.3.2 Clock Display

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

The clock is built-in, and the date (year, month, date) and time (hour, minute, second) are displayed.
Once the time is set, it can be seen as a clock on the screen.

6.3.3 Monitor Screen Display

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

A variety of information related to operation, such as the axis counter, speed display and MSTB command is displayed. The display can be switched according to the machining application (prototype or mass production).
The following operations regarding operation can be executed:

- Operation search
- Restart search
- Editing of searched machining program
- Trace (Display of machine movement path)
- Check (Display of NC program's tool movement path)
- Correction of running program's buffer
- Counter set
- Manual Numerical Value Command

etc.

■ Simple monitor screen

The information displayed on the simple monitor screen is limited. Therefore, the size of the text is larger and it is readable from a distance.

■ Selective display

A part of monitor screen can be customized to suit user's need (to display data which a user always needs to refer to).
The following can be displayed on the selective display area by selecting with the parameter:

- Tool offset
- Common variable
- Workpiece offset, etc.

Note

(1) The display is not selective on the simple monitor screen.

6.3.4 Setup Screen Display

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

Tool/workpiece related settings, user parameter settings, MDI editing, counter setting, manual numeric command issuing, etc., can be carried out.
When the data of the selected workpiece offset has been changed, the timing when the change is reflected in each counter can be selected by the parameter.

6.3.5 Edit Screen Display

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

Machining program editing (addition, deletion, change, interactive cycle insertion), program check, simple programming, playback and file input/output can be carried out.
The specified character string can be searched even during the mass-editing.

6.3.6 Diagnosis Screen Display

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

The data related CNC diagnosis can be set and displayed as follows:
- Display of hardware and software configuration
- Display of CNC options
- Diagnosis of PLC interface
- Display of drive unit information
- Display of CNC internal data
- Display of alarm message / alarm history list etc.
- Display of H/W and Operation stop status
- Setting of sampling parameter and sampling of NC internal data
- Diagnosis for functional safety

6.3.7 Maintenance Screen Display

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

Parameter setting and display, NC data input/output, NC memory format, etc., can be carried out on the Maintenance screen.
Important operations are protected with a password.

6.3.8 Home Application

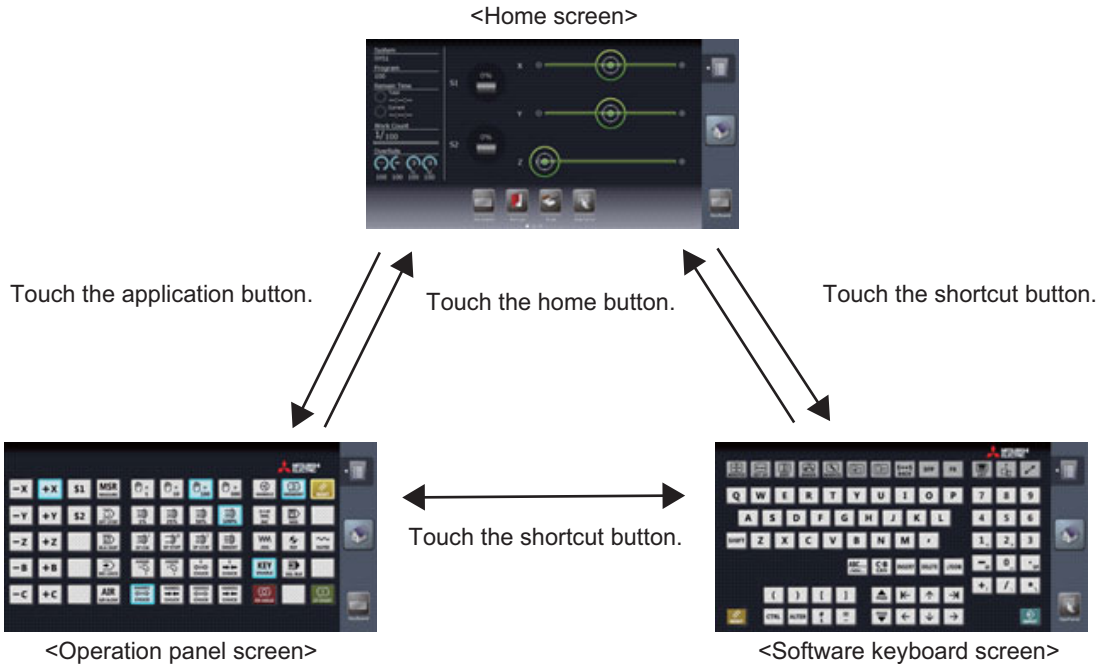
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	—	—	○	—	—
L	○	○	—	—	○	—	—

19-type vertical display unit has the expansion applications that display the machine status, software keyboard, etc. in the lower half of the screen in no linkage with the upper half.

This CNC provides the following expansion applications.

- Home screen (machine state display)
- Software keyboard
- Software operation panel
- Manual
- Notepad

MTB unique applications can also be added.

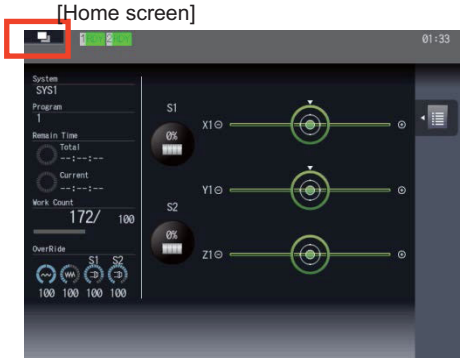


6.3.9 Home Screen

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

Home screen is able to display the machine status (including spindle loads and positions of linear and rotary axes) which can be monitored by an operator. Also, each application can be called by pressing the application button on the home screen. Registered applications differ depending on MTB.
It can be selected by the parameter as to whether to hide, display (display at power ON) or display (not display at power ON) the home screen.

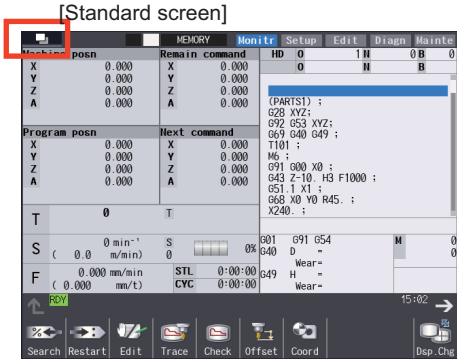
[Home screen]



Touch the standard screen changeover button

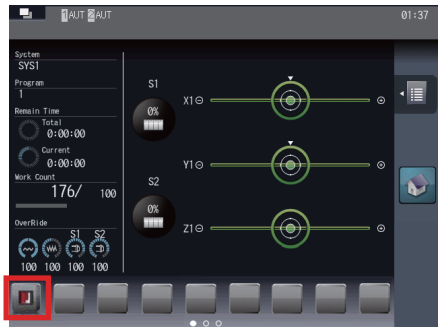
Touch the home screen changeover button

[Standard screen]



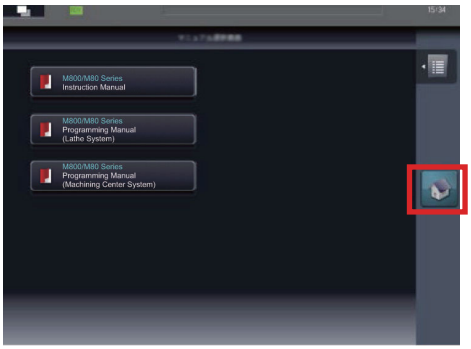
The screen can also be changed between the home screen and standard screen by pressing the window display key ( ).

(Example) Case in which you want to call the manual display application using the application button



Touch the application (manual display) button.

Touch the home button.



When the manual to be displayed is touched on the manual selection screen, the selected manual is displayed. The manuals which can be displayed differ depending on MTB.

6.3.10 Additional Languages

A language of choice can be selected by parameter setting.

6.3.10.1 Japanese

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

6.3.10.2 English

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

6.3.10.3 German

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

6.3.10.4 Italian

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

6.3.10.5 French

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

6.3.10.6 Spanish

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

6.3.10.7 Chinese

6.3.10.7.1 Chinese (Traditional Chinese Characters)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

6.3.10.7.2 Chinese (Simplified Chinese Characters)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

6.3.10.8 Korean

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

6.3.10.9 Portuguese

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

6.3.10.10 Hungarian

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

6.3.10.11 Dutch

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

6.3.10.12 Swedish

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

6.3.10.13 Turkish

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

6.3.10.14 Polish

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

6.3.10.15 Russian

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

6.3.10.16 Czech

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

6.3.10.17 Indonesian

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

6.3.10.18 Vietnamese

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

6.3.11 Display Axis Name Selection

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

This function is to switch the axis name displayed on the axis counter according to the machine axis configuration.
When the machine axis configuration is switched by arbitrary axis exchange, etc., the axis counter can be displayed with the axis name corresponding to the address commanded by the program. This makes it easier to correspond between the command and the axis counter.
Up to five axis names of the axis counter can be set with the parameters. The axis name to be displayed on the counter is switched by the system variable or PLC signal.

The axis names displayed on the following axis counters can be changed with the display axis name selection.

Screen type		Counter type
Monitor screen	Counter display	Counter display
	Selective display	Selective display area
	Workpiece coordinate system offset screen (Workpiece coordinate system offset)	Coordinate system offset area
		EXT offset area
		G92/G52 offset area
	Workpiece coordinate system offset screen (Workpiece position offset for rotary axis)	Offset amount of workpiece position for rotary axis
	Origin set screen	G92 set
	Tool center coordinate display screen	Tip machine position
		Tip workpiece position
		Counter selection
	Counter all-axis display screen	Counter all-axis display
	Counter set screen	Counter set
Setup screen	Counter display	Counter display
	Workpiece coordinate system offset screen (Workpiece coordinate system offset)	Coordinate system offset area
		EXT offset area
		G92/G52 offset area
	Workpiece coordinate system offset screen (Workpiece position offset for rotary axis)	Offset amount of workpiece position for rotary axis
	Counter set screen	Counter set

7

Input/Output Functions and Devices

7.1 Input/Output Data

Certain kinds of data handled by the NC system can be input and output between the NC system's memory and external devices.

7.1.1 Machining Program Input/Output

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

User Macros and Fixed Cycle Macros are included.

7.1.2 Tool Offset Data Input/Output

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

7.1.3 Common Variable Input/Output

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

7.1.4 Parameter Input/Output

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

7.1.5 History Data Output

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

7.1.7 System Configuration Data Output

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

7.2 Input/Output I/F

7.2.1 RS-232C I/F

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

Two ports (port 1 and port 2) can be used for control unit of the RS-232C interface.

	Control unit
Port	Port 1, Port 2
Baudrate	Up to 19.2 kbps
Handshake method	DC code method, RTS/CTS method possible

RS-232C devices can be used by connecting to each of the ports.

7.2.2 SD Card I/F

7.2.2.1 Control Unit-Side SD Card I/F [Up to 32 GB]

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	—	—	○	—	—
L	○	○	—	—	○	—	—

An SD card can be inserted into the slot inside the control unit and used.

7.2.2.2 Front-Side SD Card I/F [Up to 32 GB]

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

An SD card can be inserted into the slot in front of the display unit and used.

7.2.3 Ethernet I/F

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

The NC unit can be connected to Ethernet and used.

7.2.4 Display Unit-Side Data Server I/F

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

A built-in disk (HD) of display unit can be used.

7.2.5 Front-Side USB Memory I/F [Up to 32 GB]

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

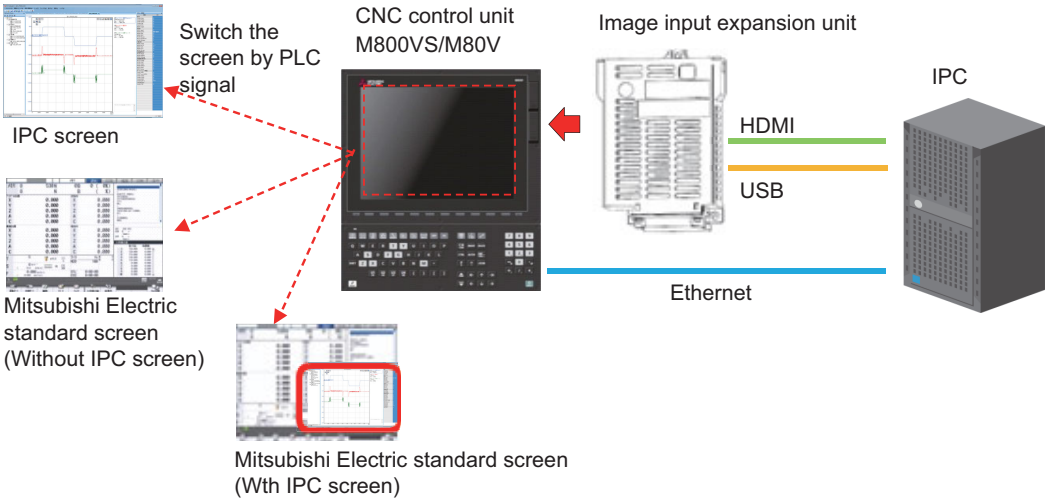
A USB memory can be mounted.

7.2.8 Image Input I/F

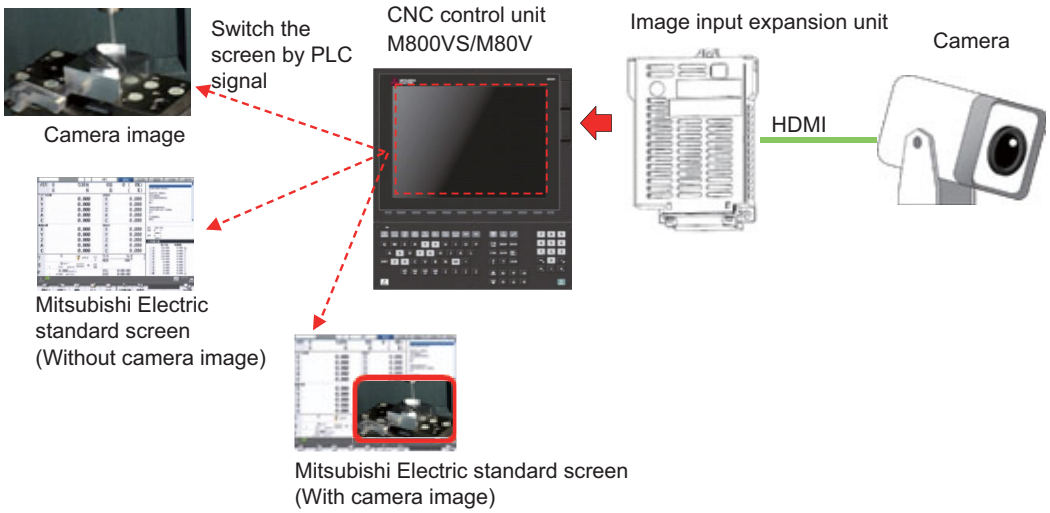
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	□	□	—	□	□
L	—	—	□	□	—	□	□

This function displays image information which was input to an image input expansion unit from an industrial PC (IPC) or a camera, on the NC display unit.
Display/non-display of the screen input from the IPC or camera image can be switched by the PLC signal or parameter setting.
The IPC screen and camera image can be partially displayed depending on the parameter setting.For the portion where the IPC screen or camera image is partially displayed, the display range and position can be changed, and the display can be enlarged/reduced and reversed vertically/horizontally.
When the IPC screen is displayed, information operated by the NC keyboard, menu key, and touchscreen is transmitted to the Mitsubishi Electric standard screen or IPC screen.
The destination to transmit information operated on the partially displayed screen (IPC screen or camera image) can be switched with the PLC signal.

[System configuration drawing (Connected with IPC)]



[System configuration drawing (Connected with camera)]



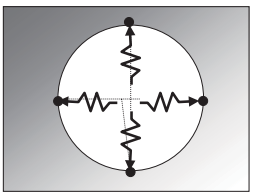
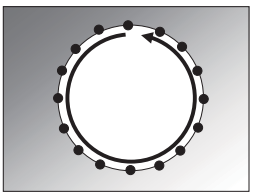
This function can be used only when the image input expansion unit is connected and the NC display unit is 10.4-type, 15-type, or 19-type.
The image input expansion unit (FCU8-EX752) is used for M8V Series. The image input expansion unit (FCU8-EX751) for M8 Series cannot be used.

7.2.9 Measuring Instrument I/F

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	—	—	—	—	—
L	□	□	—	—	—	—	—

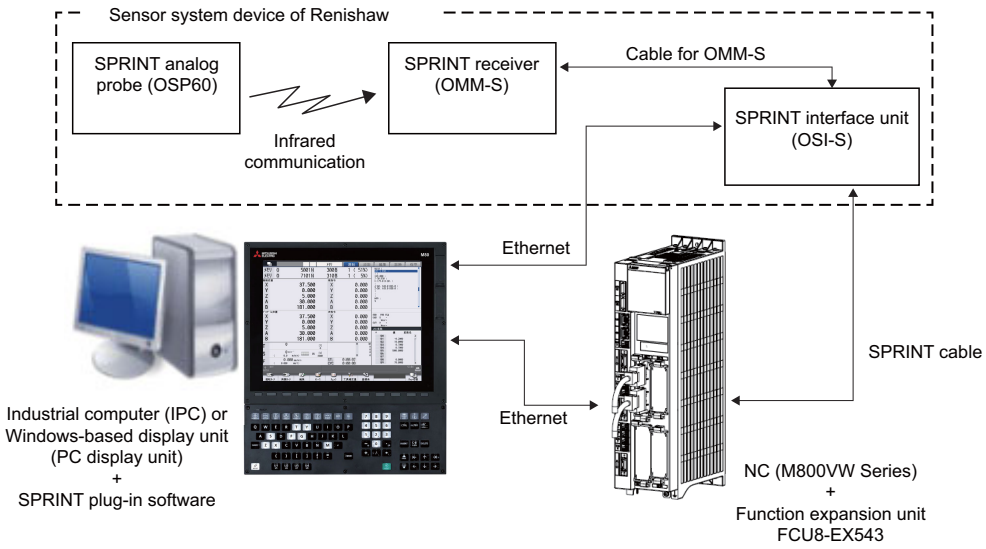
This function establishes a system such as an on-machine measurement system by connecting NC and a measuring instrument. This function enables establishment of the SPRINT system.
SPRINT is the sensor system manufactured by Renishaw, which measures the workpiece surface with a probe continuously at high speed. Compared with a method to measure the points on a workpiece one by one 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 with skip, 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 shape 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
	

The following items are required to establish the SPRINT system.
(1) M800VW Series
(2) Function expansion unit FCU8-EX543
(3) Windows-based display unit (PC display unit) or industrial computer (IPC)
(4) Sensor system of Renishaw
(5) Specifications of measuring instrument I/F of NC
(6) Cable to connect the SPRINT interface unit and M800VW Series (hereinafter called "SPRINT cable") (This needs to be prepared by MTB.)

SPRINT is configured as shown below.
[SPRINT system configuration]

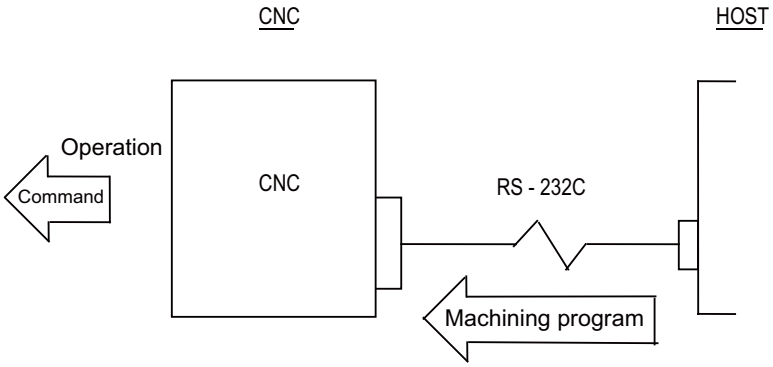


7.3 Computer Link

7.3.1 Computer Link B

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

Computer link B is a function that passes the data between host computer (hereafter abbreviated to "HOST") and CNC.
This function sends [DC1] to the HOST at the CNC cycle start, and it enables operation to be performed while the machining programs are received from the HOST.
The computer link has a reception buffer so that operation will be less susceptible to the effects of the data transfer status at the HOST end.
The high-speed machining mode function is required for high-speed micro segment machining.
This function cannot be operated in the 2nd and following part systems.

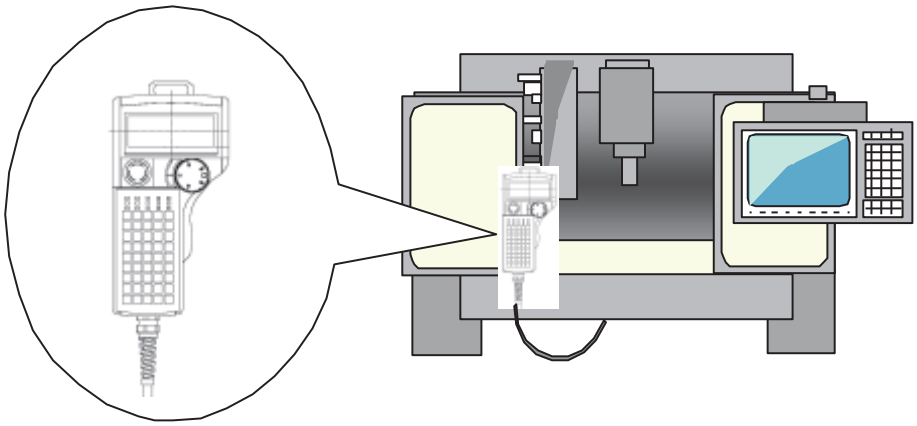


7.4 Others

7.4.1 Handy Terminal Connection

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

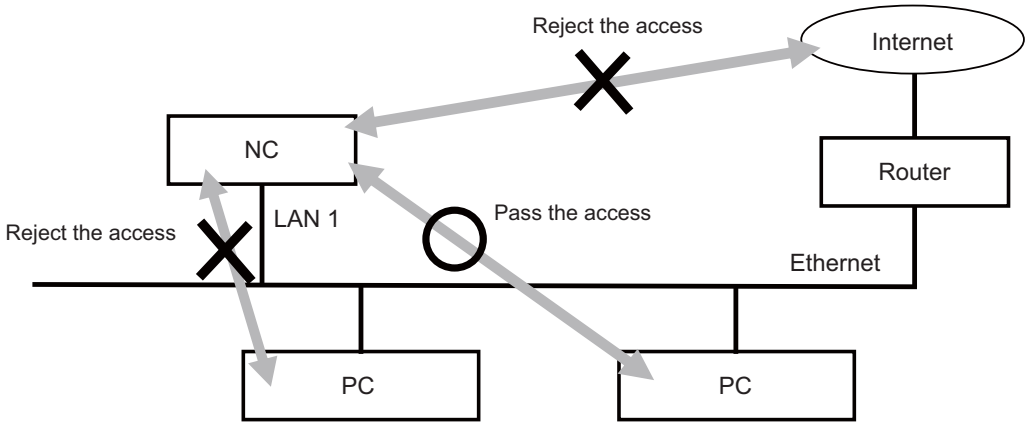
This function controls the serial communication (RS232C) of CNC and handy terminal.
Handy terminal is a downsized machine operation panel which enables you to operate the machine including setup at hand. Consequently, machine interference can be detected at hand and the usability and safety can be improved.
CNC receives data from the handy terminal and outputs them to PLC interface. CNC also sends the transmission data which PLC set in PLC interface to the handy terminal.
Each MTB needs to create PLC program corresponding to the handy terminal.
The handy terminal needs to be customized such as the configuration of the display area, key input and communication conditions with CNC.



7.4.3 IP Filter Setting

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

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.



Ports compatible with each model are as follows.
M800VW/M80VW Series: LAN1, LAN3, GDI
M800VS/M80V Series: LAN1, wireless LAN

Access to the NC is limited with one of the following two methods. This is specified by the parameter.

■ 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.

■ 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.

8

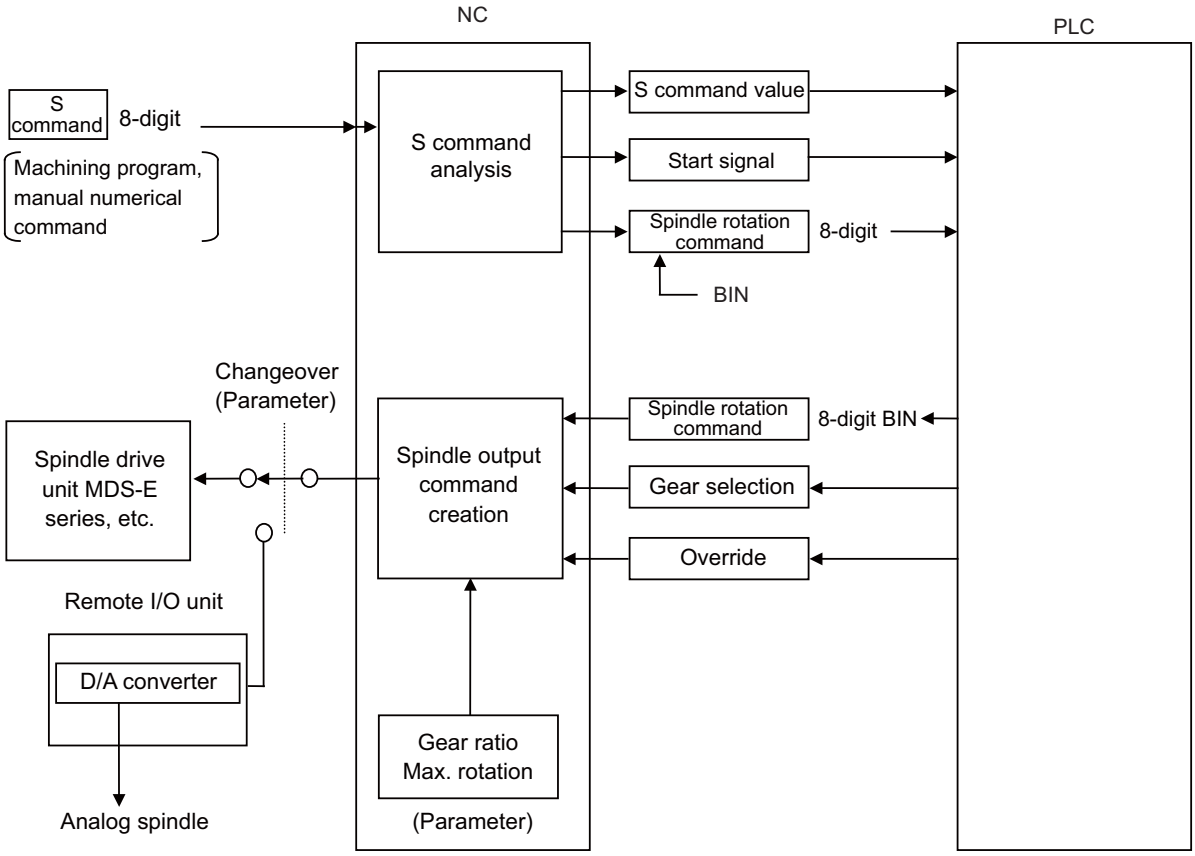
Spindle, Tool and Miscellaneous Functions

8.1 Spindle Functions (S)

8.1.1 Spindle Control Functions

The spindle rotation speed is determined in consideration of the override and gear ratio for the S command commanded in automatic operation or with manual numerical commands, and the spindle is rotated. The following diagram shows an outline of the spindle control.

When an 8-digit number following address S (S0 to S±99999999) is commanded, a signed 32-bit binary data and start signal, or a non-signed 32-bit binary data and start signal will be output to the PLC.
One set of S commands can be issued in one block.
Processing and complete sequences must be incorporated on the PLC side for all S commands.



- The override can be designated as 50% to 120% in 10% increments or 0% to 200% in 1% increments (with built-in PLC specifications).
  The override is not changed while the spindle stop input is ON, during the tapping mode, or during the thread cutting mode.
- The number of gear steps can be commanded up to four steps.
- The maximum spindle rotation speed can be set for each gear.

Note

(1) The spindle functions can work on serially connected spindles.
Check the specifications of your machine.

8.1.1.1 Spindle Digital I/F

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

This interface is used to connect the digital spindle (AC spindle motor and spindle driver).

8.1.1.2 Spindle Analog I/F

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

Spindle control can be executed using an analog spindle instead of the digital spindle.

8.1.1.3 Coil Switch

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

Constant output characteristics can be achieved across a broad spectrum down to the low-speed range by switching the spindle motor connections.
This is a system under which commands are assigned from the PLC.

8.1.1.4 Automatic Coil Switch

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

Constant output characteristics can be achieved across a broad spectrum down to the low-speed range by switching the spindle motor connections.
This is a system under which the NC unit switches the coils automatically in accordance with the motor speed.

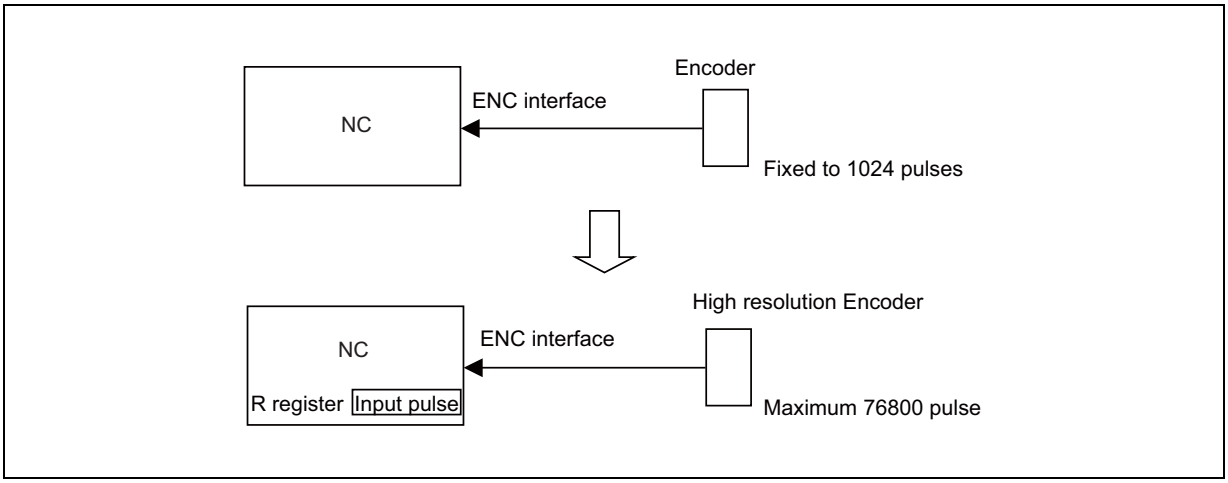
8.1.1.5 Encoder Input I/F

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	○	○	□	○	○
L	□	□	○	○	□	○	○

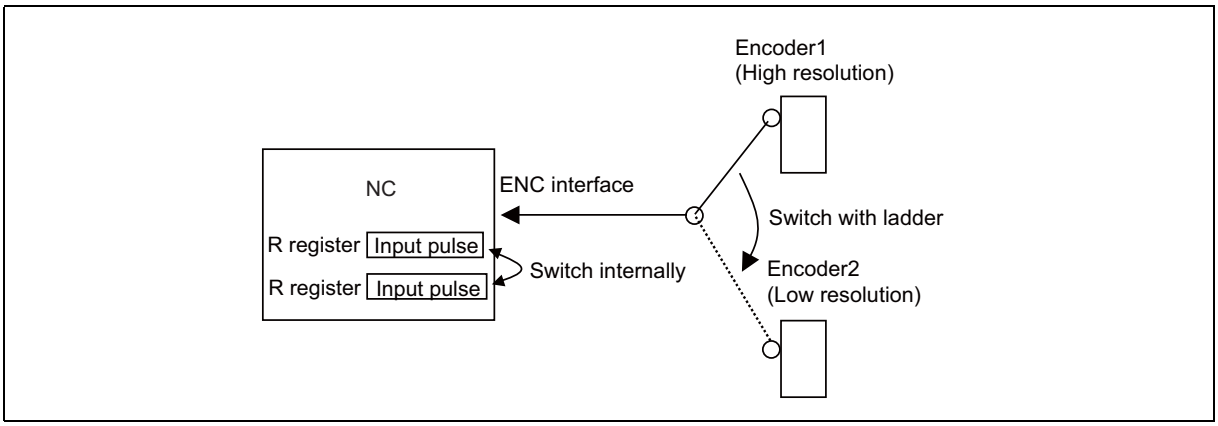
Encoder I/F expansion unit is required for M800VW/M80VW.

Arbitrary pulse can be input by values set to R register.
The maximum number of input pulse is 76800. Synchronous feed function can be activated with the arbitrary pulse if the number of pulse to be used is set in R register and the external signal is turned ON.
When the arbitrary pulse is invalid, the input pulse is fixed to 1024 pulses.

[Encoder arbitrary pulse input]



Two kinds of encoders can be switched over and connected. Encoder switch-over is possible, using PLC device.
[Encoder switched pulse input]

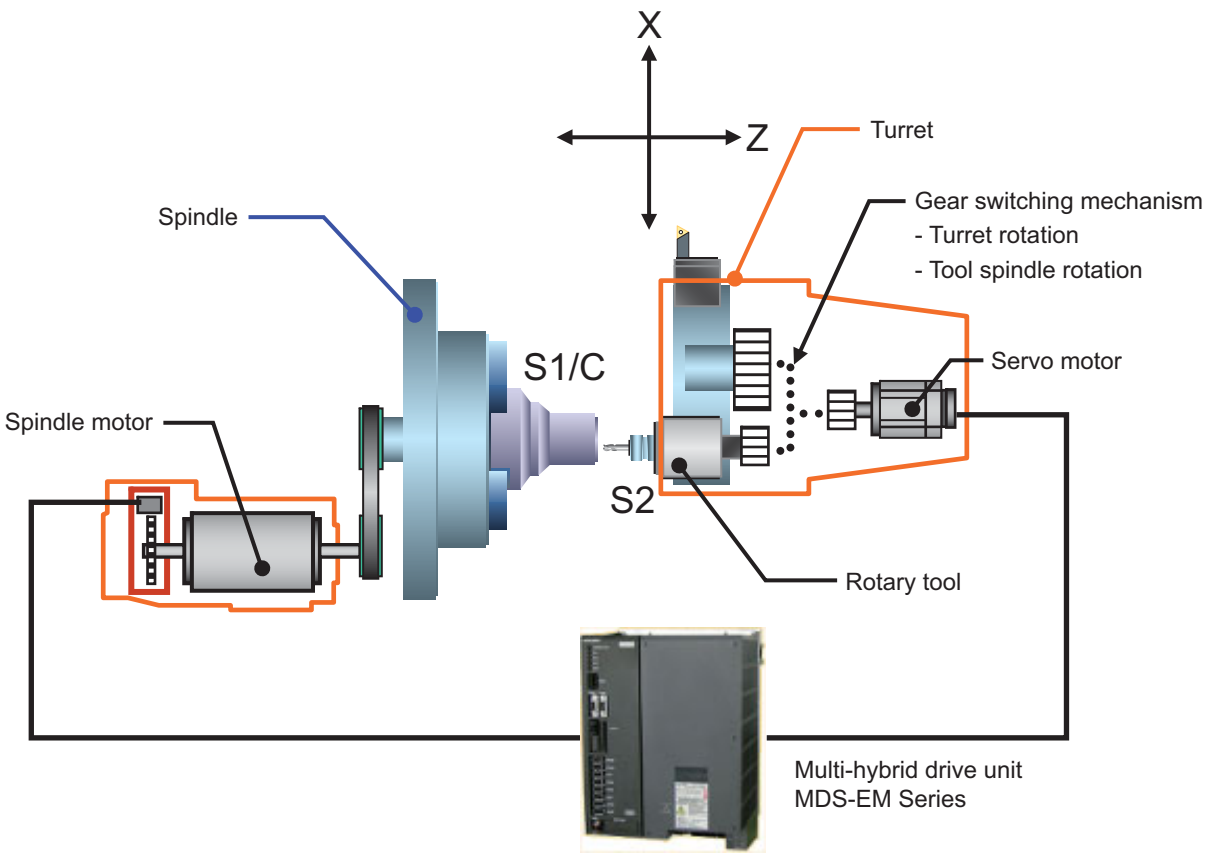


8.1.1.6 Spindle-Mode Servo Motor Control

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

This function controls a spindle using the combination of servo motor and servo drive unit which controls NC axis.
The maximum rotation speed of servo motor is low compared with a spindle motor. However, if the rated output is the same, the machine components can be downsized by using a servo motor to drive tool spindle, etc., because the servo motor is smaller than the spindle motor.
Note that some spindle functions can not be used under the spindle-mode servo motor control.

The figure below is the example of the application.
Normally, the components "spindle drive unit + spindle motor" are needed when turning, drilling or milling is performed on a lathe with turret-type tool selection mechanism. This means two pairs of "spindle drive unit + spindle motor" are needed; one for a main spindle and one for a turret.
When the spindle-mode servo motor control is used, the turret side spindle motor can be replaced with a servo motor, which realizes downsizing of the turret. When the multi-hybrid drive unit MDS-EM Series (multi-axis integrated unit with 1 spindle + 3 servo axes) is used in drive system, space-saving of the lathe and cost reduction can be realized.

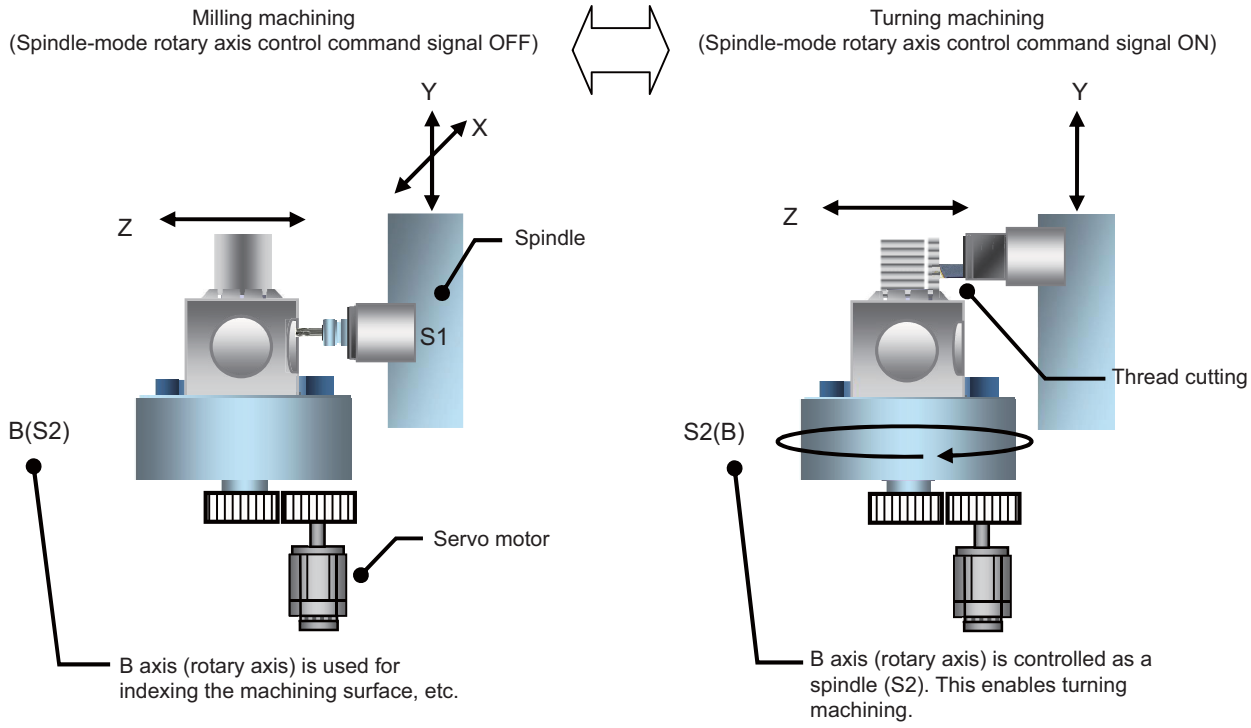


8.1.1.7 Spindle-Mode Rotary Axis Control

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	—
L	—	—	—	—	—	—	—

This function enables a rotary axis driven by a servo motor to be controlled as a spindle. When the "Spindle-mode rotary axis control command" signal is turned ON, the rotary axis is controlled as a spindle. This enables lathe-turning machining, including synchronous feed and thread cutting, to be performed in synchronization with the feedback speed of the rotary axis (spindle-mode rotary axis mode). When the signal is turned OFF, positioning and interpolation commands can be given to the rotary axis (servo axis mode).

The following is the example of the application.
Add S2 as B axis spindle. B axis can be controlled as S2 spindle by setting "2" to B axis parameter "#1020 sp_ax" and setting the same value as the B axis (servo axis) drive unit I/F number to S2 axis parameter "#3031 smcp_no". B axis can be controlled as S2 spindle by turning the "Spindle-mode rotary axis control command" signal ON after being used as B axis for indexing the machining surface. This enables turning machining, including thread cutting.



8.1.1.8 Turret Gear Change Control

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	Δ	Δ	Δ	Δ	○	○	○

This function enables axes in the semi-closed system to select four types of gear ratios which are set to the spindle specification parameters according to the control input from the PLC.

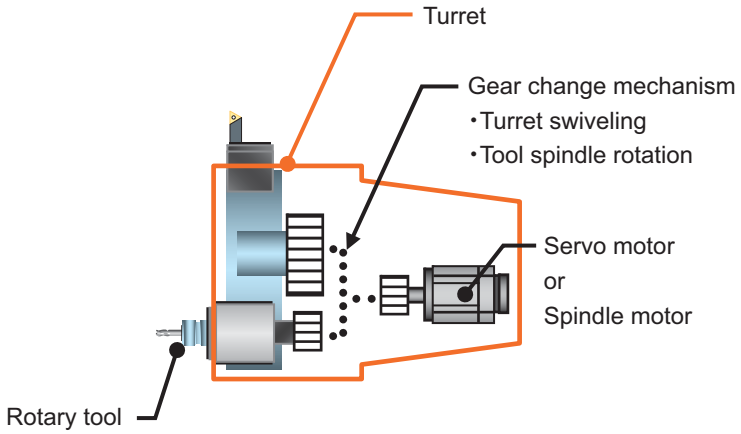
There are differences from normal gear change as follows:

- Positioning can be made at the predefined motor side position regardless of the selected gear step during spindle orientation or change to spindle position control (C axis control).
- Gears can be changed during the rotation.

■ Example of application

[Gear change between turret swiveling and tool spindle rotation in turret type tool post]

For gear change on a turret where the turret swiveling and tool rotation are switched by physical gear using one servo or spindle motor as illustrated below, the positioning to motor side Z-phase position (orientation) can be made regardless of the selected gear steps by using this function. Consequently, the gears for the physical gear change are engaged and the physical gears can be changed.



[Gear change during rotation]

For example, the tool for polygon has a tool system in which the reduction gear is for starting torque (cutter + tool folder). If this tool is installed in a tool post that has mechanism which rotates multiple tools by one motor, the gear change is needed when selecting the polygon tool. However, the gear change can be performed during the tool rotation without temporary stop so that the tool change time can be reduced.

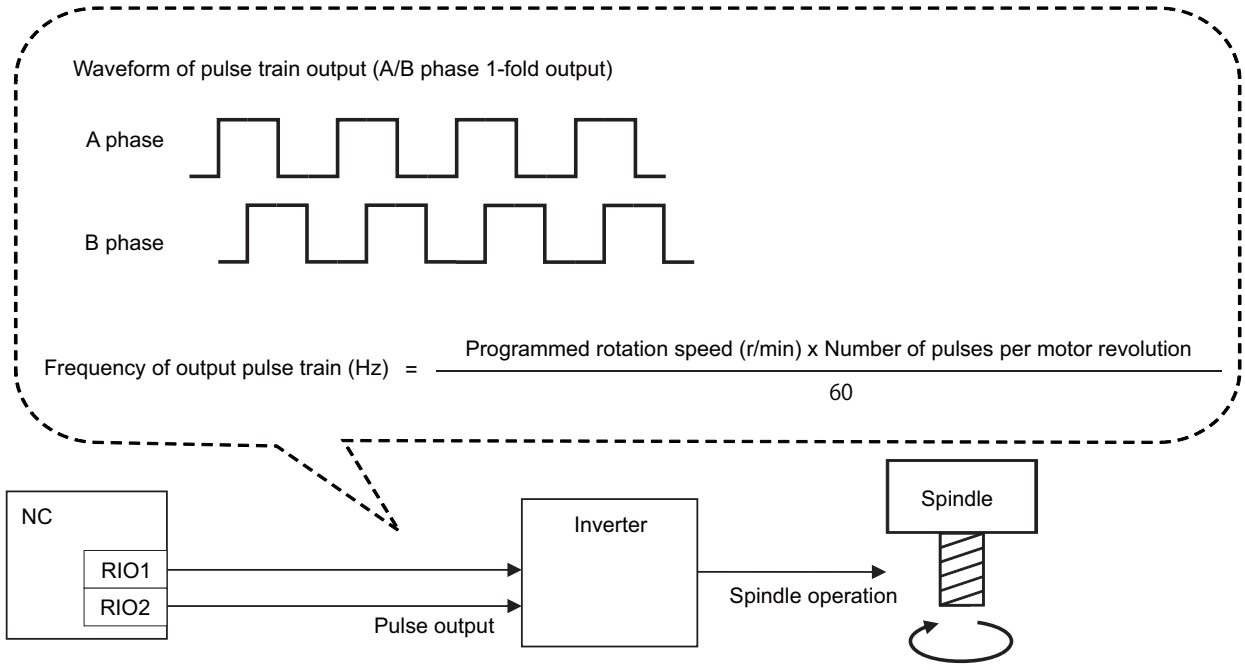
[Gear change during spindle-mode servo motor control]

This function is used when gear change is needed in the spindle-mode servo motor control which controls a servo motor as spindle using the combination of servo drive unit and servo motor.

8.1.1.9 Spindle Control with Pulse Train Output

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	O	O
L	—	—	—	—	—	O	O

This function controls a spindle using pulse trains output from the NC. You can control a spindle which is connected mainly to an inverter by outputting pulse trains.
Using this function requires an inverter which is capable of controlling spindle operation through pulse train input.
When this function is used, RIO port 1 and 2 are exclusively used for outputting pulse trains.



8.1.2 S Code Output

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

When an 8-digit number following address S (S0 to S±99999999) is commanded, a signed 32-bit binary data and start signal, or a non-signed 32-bit binary data and start signal will be output to the PLC.
One set of S commands can be issued in one block.
Processing and complete sequences are required on the PLC side for all S commands.

S function can be designated with any other kind of commands. In the case where a movement command is in the same block, two different command sequences are available. Depending on user PLC specifications, either one of the following two will be applied.

- S function is executed after the movement is completed.
- S function is executed at the same time as when the movement command is issued.

8.1.3 Constant Surface Speed Control

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

This function enables the spindle rotation speed to be adjusted (constant surface speed control) in accordance with the movement of tool nose point so that the cutting point always remains at the constant speed (constant surface speed).
Using this function for processes such as a cutting-off process is effective for a machining time and tool life, etc.
Note that when the tool nose point is moving to the workpiece zero point, the rotation may be at the maximum rotation speed of the machine specification and this is dangerous. Be sure to specify the maximum clamp rotation speed with the spindle clamp speed setting command (G92/G50).

■ Command format

Constant surface speed ON

G96 S__ P__ ;		
S	:	Surface speed designation (-99999999 to 99999999 (m/min), (feet/min))
P	:	Constant surface speed control axis designation (0 to the maximum number of the controllable axes in G96 part system.)

Constant surface speed cancel

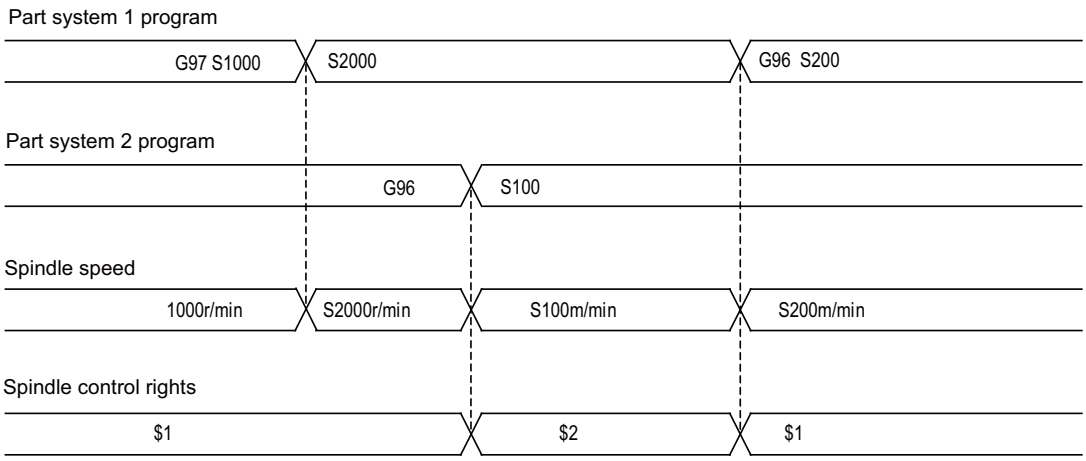
G97 S__ ;		
S	:	Spindle rotation speed (-99999999 to 99999999 (r/min))

The surface speed is commanded with an S code. For the metric designation, the speed is commanded with an m/min unit, and for the inch designation, the speed is commanded with a feet/min unit.
In the constant surface speed cancel mode, the S code is a spindle rotation speed command.
The axis for which constant surface speed is controlled is generally the X axis. However, this can be changed with the parameter settings or with address P in the G96 block.
Select with the parameter whether to enable the spindle rotation command from other part systems to the spindle which is in the constant surface speed control.

Note

(1) 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 or may cause damage to the operators. Thus make sure to use this control while the "spindle speed clamp" is enabled. When the constant surface speed control is commanded, keep enough distance from the program zero point.

(2) If there is only one spindle, the spindle will not operate normally if the constant surface speed control command, S command or spindle related M command is commanded randomly from each part system. These commands must be commanded from only one certain part system, or commanded simultaneously with timing synchronization function.
The controller will execute the following control for the constant surface speed control and S commands. The part system from which an S command was issued last will have the spindle control rights. That part system will judge whether the constant surface speed command mode is valid or canceled, and will execute spindle control.



$1: Part system 1
$2: Part system 2

8.1.4 Spindle Override

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

This function applies override to the rotation speed of a spindle or mill spindle assigned by the machining program command during automatic operation or by manual operation. There are two types of override.

■ Type 1 (code method)

Using an external signal, override can be applied to the commanded rotation speed of a spindle or mill spindle in 10% increments from 50% to 120%.

■ Type 2 (value setting method)

Using an external signal, override can be applied to the commanded rotation speed of a spindle or mill spindle in 1% increments from 0% to 200%.

Note

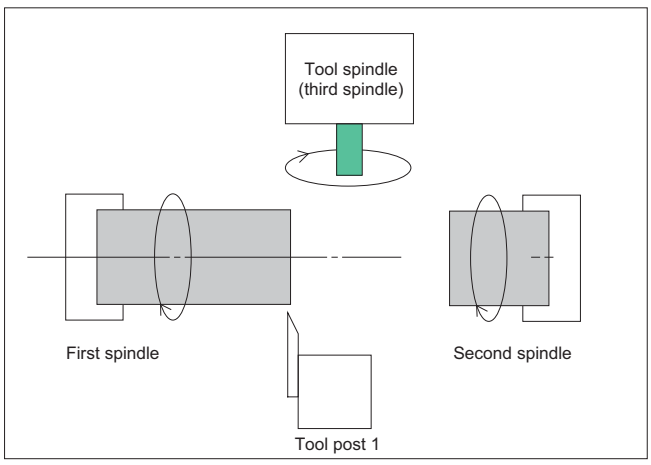
(1) Selection between type 1 and type 2 can be designated by user PLC processing.

8.1.5 Multiple-Spindle Control

Multiple-spindle control is a function that controls second and following spindles in addition to the first spindle in a machine tool equipped with multiple spindles.
Multiple-spindle control I can be switched to multiple-spindle control II or vice versa using a parameter and, by so doing, the spindle control method changes.

Multiple-spindle control I (L system only)	:	Control based on a spindle selection command (such as G43.1) and spindle control command ([S*****;] or [SO=*****;]), etc.
Multiple-spindle control II	:	Control based on a PLC signal ("Spindle command selection" signal, "Spindle selection" signal) and spindle control command ([S*****;] only), etc. Spindle selection command and control command of [SO=*****;] cannot be used.

The figure below shows an example of the configuration for a machine which is equipped with second and third spindles.



8.1.5.1 Multiple-Spindle Control I

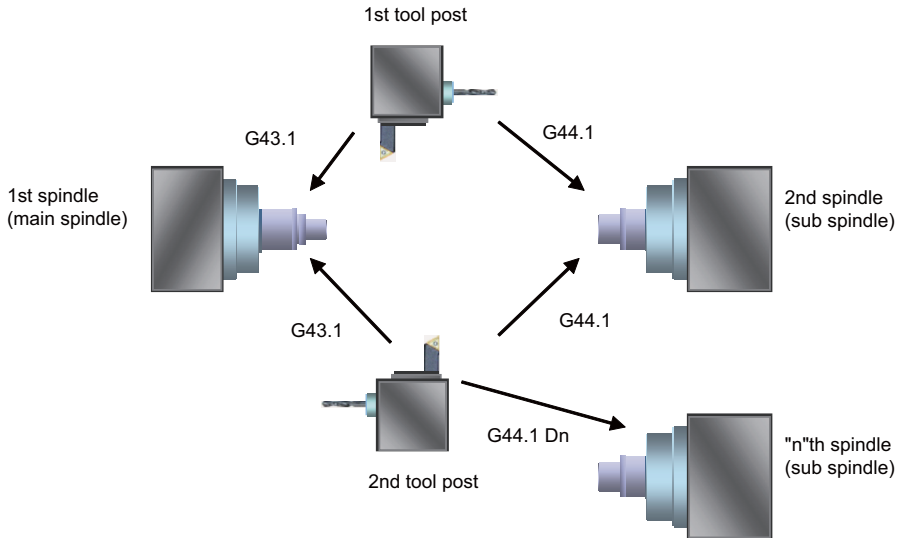
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	O	O	O	O	O	O	O

Spindle selection command

In a machine with two or more spindles, this function makes it possible to select the spindle to apply the S commands or feed per revolution.
Spindle selection command (G43.1/G44.1/G44.1 D_) makes it possible to select the spindle. The spindle also can be selected with the P address in the same block as the S command depending on the parameter setting.

Using the 1st spindle control command (G43.1) and the 2nd spindle control command (G44.1), the commands can be given to the spindle designated with the parameter.
Using the arbitrary spindle control command (G44.1 D_), an arbitrary spindle can be selected with address D.
Using all spindles simultaneous control command (G47.1), all spindles are selected.
Use this function when cutting the back of workpiece by shifting it to the second spindle from the first spindle, cutting multiple workpieces with the respective spindles or on the respective tool posts, etc.

[When selecting G43.1 = 1st spindle, G44.1 = 2nd spindle]



■ Command format

G43.1 ;	**First spindle control mode ON (The spindle to be used is set using the parameter.)**

G44.1 ;	**Second spindle control mode ON (The spindle to be used is set using the parameter.)**

G44.1 D__;	**Arbitrary spindle control mode ON (The spindle to be used is set using address D.)**

G47.1 ;	**All spindles simultaneous control mode ON (Encoder of the 1st spindle is used)**

D	Designate the spindle. (For a spindle number command, the range is from 1 to the number of available spindles, or from 1 to 9 for a spindle name command.)

Spindle selection with address P

The spindle selection function is enabled with the P address in the same block as the M3/M4/M5 command depending on the parameter setting.

The S, M and P commands can be issued in random order.

```
S__ P__ ;
```

```
M__ P__ ;
```

```
S__ M__ P__ ;
```

S	Rotation speed or surface speed command value
M	Spindle command (forward rotation/reverse rotation/stop)
P	Spindle selection code

Spindle control command (Using extended word address (S○=*****))

In addition to using the "S*****" command, using the "S○=*****" command makes it possible to assign commands which differentiate each spindle.

■ Command format

```
S ○ =***** ;
```

○	Designate the spindle No. with one numeric character (1: the first spindle/2: the second spindle/ .../n: the "n"th spindle (n = maximum number of spindles for the specification)). Variables can be designated.
*****	Rotational speed or surface speed value assigned by command. Variables can be designated.

8.1.5.2 Multiple-Spindle Control II

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

This function selects which spindle to perform the S command or feed per revolution (synchronous feed) on a machine equipped with 2 or more spindles.

Command to spindles is issued with one S command, and you can select the target spindle with a PLC signal (spindle selection or spindle command selection).

Depending on combination of the parameters, when the "Spindle command selection" signal is changed in the same block as the S command, the S command is applied to both spindles before and after the change.

A parameter is used to switch the control to multiple-spindle control I.

There are two types of spindle rotation speed control: common to part systems (which validates the S command issued last to each spindle), and separated by each part system (which validates the S command issued last in the part system to which the selected spindle belongs).

8.1.6 Spindle Orientation

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

This function stops the spindle at a specified position.
The spindle zero position is determined to be the Z-phase position at the encoder method spindle orientation (PLG and external encoder). At the proximity switch method spindle orientation, the spindle zero position is determined to be the position of the proximity switch attached at the spindle end.
On a lathe, this function can be used for the workpiece positioning to align the attaching/removing direction and the turret turn positioning for the tool exchanges.
When Z-phase has not yet been passed, pass Z-phase twice (excluding reciprocations within one rotation), and then position to the orientation position.
When Z-phase has been passed already, immediately position to the orientation position.
The rotation direction of the spindle orientation is determined by the spindle zero point return specification parameter.
The spindle does not rotate if Z-phase has been passed and the orientation position is already established when orientation command is issued.
When the spindle has been rotating in the direction opposite from that of orientation rotation when the orientation command is issued, orientation operation will be executed after decelerating to stop. By using the parameter, the orientation operation can be executed with the spindle revolution direction as it is, without having to decelerate and stop.

■ Orientation

This function stops the spindle rotation at a certain position when using the digital spindle.
When the orientation command is used, the spindle will rotate several times and then stop at the orientation point.
The orientation position differs depending on the detector.

- When the orientation (PLG and external encoder) is used:
  At the Z-phase position
- When the proximity switch method orientation is used:
  At the proximity switch installation position

■ Multi-point orientation

This function performs orientation to a position other than the Z-phase position by inputting a shift amount with the parameter or PLC. The shift amount is -35999 to 35999. (Unit: 360°/36000=0.01°)

Note

(1) Multi-point orientation cannot be executed when using the proximity switch.
(2) Orientation is possible only when the gear ratio is 1:1 for the PLG orientation.
(The orientation is completed at the PLG encoder's Z-phase, so when using reduction gears, the orientation points will be generated at several points during one spindle rotation.)

■ Orientation in-position advance output

This function turns the "Spindle in-position" signal ON as soon as the spindle reaches within the second in-position width. Then, the "Spindle 2nd in-position" signal is turned ON as soon as the spindle reaches within the in-position width.
Since orientation completion can be predicted using this function, it is possible to eliminate the sequence delay time, etc. for tool changes and other such operations, thereby achieving a faster takt time.

■ Proximity switch method orientation

Proximity switch method orientation enables the spindle positioning by providing the proximity switch position installed on the spindle side as the spindle position zero point.
When the spindle motor and the spindle are connected with V-belt, the spindle position zero point calculated from the spindle motor position varies due to belt slipping, etc. Thus, the spindle position zero point must be detected from the proximity switch signal every time orientation or zero point return is carried out.
In the conventional spindle position zero point detection method, the spindle position zero point is calculated from the spindle drive unit and transmitted to NC when the position detector detects the one-time rotation signal (Z-phase) for the first time after the system has been turned ON, and the zero point will not be changed thereafter.
In the case of proximity switch signal orientation, NC updates the spindle position zero point every time orientation or zero point return is carried out.

8.1.7 Spindle Position Control (Spindle/C Axis Control)

8.1.7.1 Spindle Position Control (Spindle/C Axis Control)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

A spindle can be controlled as the rotary axis. After switching the spindle to the rotary axis, the positioning and the interpolation between the spindle and other NC axes can be operated in the same way as the NC axis by executing the position command (the movement command).
The NC axis for controlling the spindle as the rotary axis or the machinery for switching the spindle and NC axis (such as a gear switching machinery) had been necessary for controlling a spindle readily as the rotary axis, but they are not necessary with this function.
There are two methods for switching a spindle and a rotary axis: PLC signal method and program command method. The method can be selected with the parameter.

PLC signal method

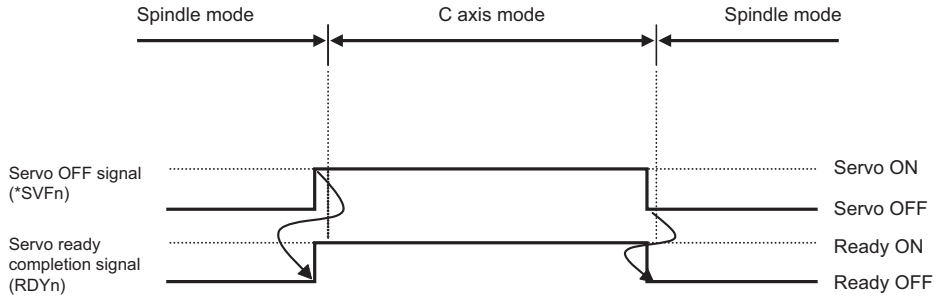
The spindle mode and the C axis mode can be switched with the "Servo OFF" signal (*SVFn) ON/OF with PLC ladder.
The interpolation operation is enabled by executing the movement command to the C axis in the C axis mode same as NC axis.

[Switching spindle mode to C axis mode]

Change the "Servo OFF" signal (*SVFn) OFF to ON.

[Switching C axis mode to spindle mode]

Change the "Servo OFF" signal (*SVFn) ON to OFF.



Program command method

In the machining program, the program switches to the C axis mode with G00 command, and to the spindle mode with S command. The C axis "Servo OFF" signal (*SVFn) must be kept ON while the program command method is selected.

[Switching spindle mode to C axis mode]

Command [G00 C__] in the NC program during the spindle mode. The axis is positioned directly to the specified position. The non-interpolation positioning for each axes is performed by specifying [G00 X__ Z__ C__] regardless of the parameter setting, and C axis is switched to the C axis mode.

[Switching C axis mode to spindle mode]

- The switching is performed with the spindle forward run signal (SRN) ON or the spindle reverse run signal (SRI) ON and the S command.
- The switching is performed with the rising edge of the spindle forward run signal (SRN) or the spindle reverse run signal (SRI).

8.1.7.2 C Axis Control During Spindle Synchronization

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	—	—	—
L	Δ	Δ	Δ	Δ	○	○	○

This control enables C axis positioning while a long workpiece is controlled by front and rear spindles that are in synchronization with each other. Under this control, the machine can perform milling at the center of workpiece.

8.1.8 Spindle Synchronization

In a machine with two or more spindles, this function controls the rotation speed and phase of one selected spindle (synchronized spindle) in synchronization with the rotation of the other selected spindle (reference spindle). This function can be assigned by G code or by PLC.
It is used in cases where, for instance, workpiece clamped to the reference spindle is to be clamped to the synchronized spindle instead or where the spindle rotation speed is to be changed while one workpiece remains clamped to both spindles.

8.1.8.1 Spindle Synchronization I

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

The synchronized spindle is designated and the start/end of the synchronization are commanded with the G command in the machining program.

This function cannot be combined with the following spindle synchronization functions while it is active. An operation error occurs.
- Spindle synchronization I
- Spindle synchronization II
- Tool spindle synchronization IA
- Tool spindle synchronization IB (IC)
- Tool Spindle Synchronization II
- Spindle superimposition control

However, if multiple spindle synchronization set control is enabled, multiple spindle synchronization functions can be commanded. For details on the multiple spindle synchronization set control, refer to " 8.1.15 Multiple Spindle Synchronization Set Control".

■ Command format

Spindle synchronization control ON (G114.1)
This command is used to designate the reference spindle and the spindle to be synchronized with the reference spindle, and it places the two designated spindles in the synchronized state.
By designating the synchronized spindle phase shift amount, the phases of the reference spindle and synchronized spindle can be aligned.

G114.1 H__ D__ R__ A__ ;

H	Designate the reference spindle.
D	Designate the spindle to be synchronized with the reference spindle.
R	Designate the synchronized spindle phase shift amount.
A	Designates the spindle synchronization acceleration/deceleration time constant.

Spindle synchronization control cancel (G113 or G113.1)
This command releases the state of synchronization between two spindles whose rotation has been synchronized by the spindle synchronization command.

G113 ; **(Lathe system)**

G113.1 ; **(Lathe system/Machining center system)**

8.1.8.2 Spindle Synchronization II

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

The selection of the spindles to be synchronized, the start of the synchronization and other settings are all designated from the PLC.

The spindle synchronization control mode is established by inputting the "Spindle synchronization" signal. While this mode is established, the synchronized spindle is controlled in synchronization with the rotation speed assigned for the reference spindle.

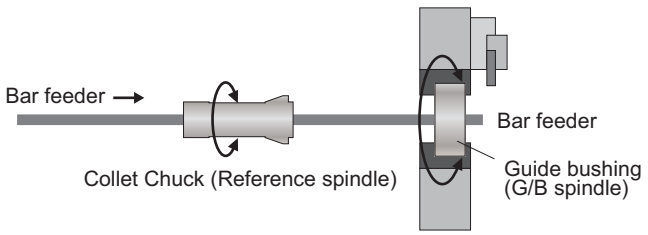
This function cannot be combined with the following spindle synchronization functions while it is active. An operation error occurs.

- Spindle synchronization I
- Spindle synchronization II
- Tool spindle synchronization IA
- Tool spindle synchronization IB (IC)
- Tool Spindle Synchronization II
- Spindle superimposition control

8.1.8.3 Guide Bushing Spindle Synchronization

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	Δ	Δ	Δ	Δ	○	○	—

This function is for a machine with a spindle motor to rotate a guide bushing: This function allows the guide bushing spindle motor (G/B spindle) to synchronize with a reference spindle motor (Reference spindle). The position error compensation function reduces the spindle's vibration due to the workpiece's torsion, and the motor's overload.



8.1.9 Tool Spindle Synchronization I (Polygon)

8.1.9.1 Tool Spindle Synchronization IA (Spindle-Spindle, Polygon)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

In a machine containing a rotary tool axis and that has a spindle controlled as the workpiece axis, spindle-spindle polygon machining can be carried out by controlling the workpiece axis rotation in synchronization with the rotation of the rotary tool axis.

This function can be used with the G code list 1 to 5.

This function cannot be combined with the following spindle synchronization functions while it is active. An operation error occurs.
- Spindle synchronization I
- Spindle synchronization II
- Tool spindle synchronization IA
- Tool spindle synchronization IB (IC)
- Tool Spindle Synchronization II
- Spindle superimposition control

However, if multiple spindle synchronization set control is enabled, multiple spindle synchronization functions can be commanded.

■ Command format

Tool spindle synchronization IA mode command (G114.2)
This command establishes the spindle-spindle polygon machining mode (IA) in which two spindles are rotated in synchronization at two different speeds. It is required to designate the rotary tool spindle, workpiece spindle and the rotational ratios (number of rotary tool teeth and number of work angles) of the two designated spindles.

G114.2 H__ D__ E__ L__ R__ ;	
H	: Selects the rotary tool spindle (spindle).
D	: Selects the workpiece spindle (spindle).
E	: Designates the rotary tool spindle rotational ratio.
L	: Designates the workpiece spindle rotational ratio.
R	: Synchronized spindle phase shift amount

Tool spindle synchronization IA mode cancel command (G113/G113.1)
This command releases the state of synchronization between two spindles whose rotation has been synchronized by the spindle synchronization command.

G113 ;	**(Lathe system)**

G113.1 ;	**(Lathe system/Machining center system)**

Note

(1) An axis that involves any travel cannot be put in the same block as Tool spindle synchronization IA mode cancel command. If the axis address is included in the block, a program error occurs upon the cancel command, causing the automatic operation to stop.

8.1.9.2 Tool Spindle Synchronization IB (Spindle-Spindle, Polygon)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

In a machine containing a workpiece axis and that has a spindle controlled as the rotary tool axis, spindle-spindle polygon machining can be carried out by controlling the rotary tool axis rotation in synchronization with the rotation of the workpiece axis.
A parameter is used to switch from Tool spindle synchronization IB to Tool spindle synchronization IC or vice versa.

This function can be used with the G code list 1, 6 and 7.

This function cannot be combined with the following spindle synchronization functions while it is active. An operation error occurs.
- Spindle synchronization I
- Spindle synchronization II
- Tool spindle synchronization IA
- Tool spindle synchronization IB (IC)
- Tool Spindle Synchronization II
- Spindle superimposition control

However, if multiple spindle synchronization set control is enabled, multiple spindle synchronization functions can be commanded.

■ Command format

Tool spindle synchronization IB mode ON (G51.2 or G251)
This command establishes the spindle-spindle polygon machining mode in which two spindles are rotated in synchronization at two different speeds. It is required to designate the rotary tool spindle, workpiece spindle and the rotational ratios (number of work angles and number of rotary tool teeth) of the two designated spindles.

G51.2 H__ D__ P__ Q__ R__ ;	
H	: Selects the workpiece spindle (spindle).
D	: Selects the rotary tool spindle (spindle).
P	: Designates the workpiece spindle rotational ratio.
Q	: Designates the rotary tool spindle rotational ratio.
R	: Synchronized spindle phase shift amount

Tool spindle synchronization IB mode cancel command (G50.2 or G250)

G50.2 ;

Note

(1) An axis that involves any travel cannot be put in the same block as Tool spindle synchronization IA mode cancel command. Doing so causes a program error when the spindle synchronization IB mode cancel command is issued, which causes automatic operation to be paused.

8.1.9.3 Tool Spindle Synchronization IC (Spindle-NC Axis, Polygon)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	—
L	Δ	Δ	Δ	Δ	○	○	—

This function controls so that the workpiece (spindle) and tool (NC axis) synchronously rotate at the commanded ratio and allows polygon machining. By changing the rotational ratio of workpieces and tools and the number of tool blades mounted, polygons such as rectangles and hexagons can be machined.
A parameter is used to switch from Tool spindle synchronization IB to Tool spindle synchronization IC or vice versa.

This function can be used with the G code list 1, 6 and 7.

■ Command format

Tool spindle synchronization IC mode ON (G51.2 or G251)

G51.2 P__ Q__ ;	**Spindle synchronization start (Tool spindle synchronization IC mode start)**

P	Spindle rotational ratio
Q	Rotary tool axis rotational ratio Rotation direction is specified with a sign. +: Forward rotation, -: Reverse rotation

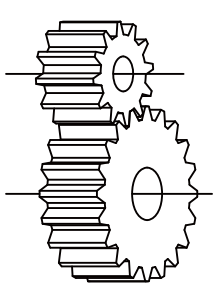
Tool spindle synchronization IC mode OFF (G50.2 or G250)

G50.2 ;

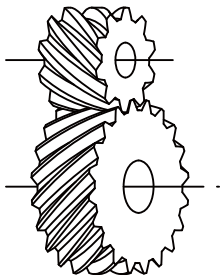
8.1.10 Tool Spindle Synchronization II (Hobbing)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	△	△	△	△	○	○	—
L	△	△	△	△	○	○	—

This function is to cut the gear with hob (hob cutter).
A spur gear can be machined by synchronizing and rotating the hob axis and the workpiece axis in a constant ratio.
A helical gear can be machined by compensating the workpiece axis according to the gear torsion angle for the Z axis movement.
Different command format is used for G code list 1 to 5 and G code list 6 or 7.

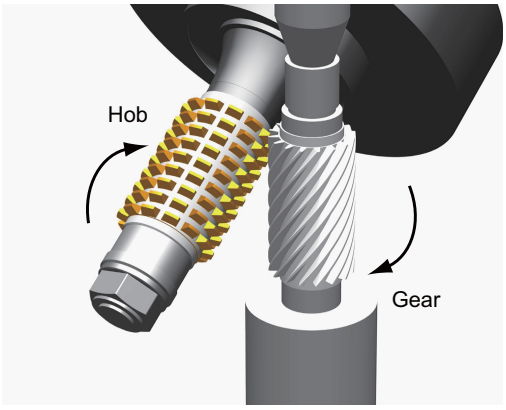


Spur gears



Helical gears

By synchronizing and rotating the hob axis and the workpiece axis in a constant rotation ratio, a gear is machined so that the cutter is engaged with gear.



Hob axis: Rotary tool spindle to mount a hob
Workpiece axis: Rotary axis to mount a workpiece

This function can be used in the following system according to the command format.
- G114.3 command: L system/M system
- G81.4 command: L system

This function cannot be combined with the following spindle synchronization functions while it is active. An operation error occurs.
- Spindle synchronization I
- Spindle synchronization II
- Tool spindle synchronization IA
- Tool spindle synchronization IB (IC)
- Tool Spindle Synchronization II
- Spindle superimposition control

However, if multiple spindle synchronization set control is enabled, multiple spindle synchronization functions can be commanded.

8.1.11 Spindle Speed Clamp

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

This function is to limit the spindle rotation speed to the ranges between the maximum rotation speed and the minimum rotation speed.
This function can be set by a parameter or a program.

G92 S__ Q__ ;	**Spindle speed clamp setting**

S	Maximum clamp rotation speed
Q	Minimum clamp rotation speed

An alarm will occur if the constant surface speed control is conducted without issuing the spindle clamp speed command (G92).

8.1.13 Spindle Oscillation

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	—	—	—
L	Δ	Δ	Δ	Δ	—	—	—

This function reciprocates (oscillates) the spindles with designated amplitude and frequency.

8.1.14 Spindle Superimposition Control

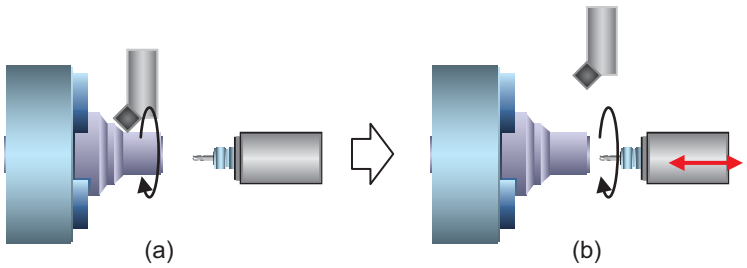
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	Δ	Δ	Δ	Δ	○	○	—

Spindles are controlled by superimposing the rotation speed of one spindle on the rotary speed of other spindle.
Use this function when the tool spindle needs to be rotated with the superimposed speed on the spindle rotation speed. If this function is used when tapping is performed to the center of a workpiece clamped to a spindle during its rotation, for instance, the tapping to the center of the workpiece (shown in Figure (b) below) can be performed during the turning machining (shown in Figure (a) below) so that the cycle time can be reduced.

This function cannot be combined with the following spindle synchronization functions while it is active. An operation error occurs.
- Spindle synchronization I
- Spindle synchronization II
- Tool spindle synchronization IA
- Tool spindle synchronization IB (IC)
- Tool Spindle Synchronization II
- Spindle superimposition control

However, if multiple spindle synchronization set control is enabled, multiple spindle synchronization functions can be commanded.

■ Conventional method



Synchronous tapping
Forward:
+500 (r/min)
Backward:
-500 (r/min)

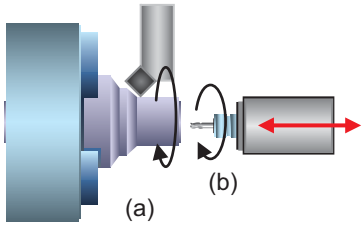
S1=1000(r/min)

S1 = 0 (r/min)

Normal turning machining

The rotation of the front workpiece is stopped, and then the tapping is started at the front center.

■ Using the spindle superimposition control



Synchronous tapping
Forward:
+1500 (r/min)
Backward:
+500 (r/min)

S1=1000(r/min)

Turning (a) and tapping at the front center (b) are performed simultaneously.

■ Command format

Spindle superimposition control valid command

G164 command specifies the reference spindle and superimposed spindle and makes the two spindles to be superimposed.

G164 H__ D__ ;

H : Select the reference spindle (For command using the spindle number, the spindle number should be within the number of spindles for specifications. For command using the spindle name, the name is valid from 1 to 9.)

D : Select the superimposed spindle (For command using the spindle number, the spindle number should be within the number of spindles for specifications. For command using the spindle name, the number from 1 to 9 or from -1 to -9 is valid for the spindle name.)

Command with the spindle name if all spindle names are set to the spindle name parameter. For others, command with the spindle number.

Spindle superimposition control cancel command

G113 command cancels the superimposition of two spindles which are rotating by the differential velocity rotation tool command.

The superimposition can be cancelled also with "Spindle synchronization cancel" signal ON.

G113 ;

G113.1 ;

For G code list 6 and 7, only G113.1 is available for the cancel command.

Note

(1) An axis that involves any travel cannot be put in the same block as the spindle superimposition control cancel command. If the axis address is included in the block, a program error occurs upon the cancel command, causing the automatic operation to stop.

8.1.15 Multiple Spindle Synchronization Set Control

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	O	O	O	O	O	O	O

This function enables users to combine the following spindle control functions and control them simultaneously.
The same function can be combined.
When a certain pair of spindles has been designated for one spindle control function, a different pair of spindles can be used for another spindle control function.

- Spindle synchronization I
- Tool spindle synchronization I A (Spindle-Spindle, Polygon)
- Tool spindle synchronization I B (Spindle-Spindle, Polygon)
- Tool spindle synchronization II (Hobbing)
- Spindle superimposition control

The following spindle control functions can be commanded simultaneously by using the same reference spindle.

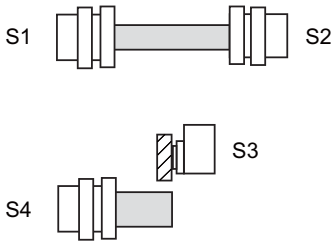
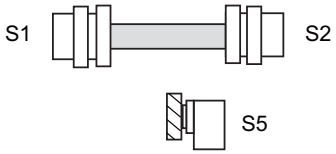
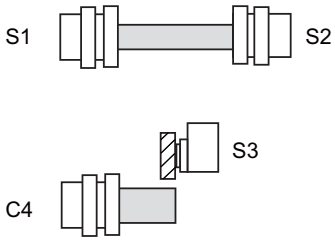
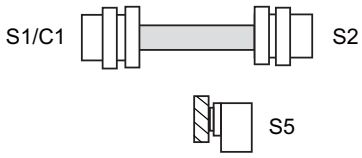
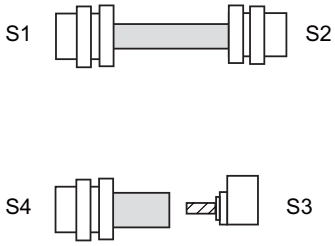
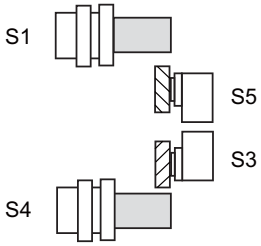
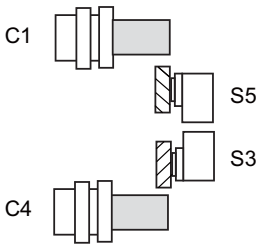
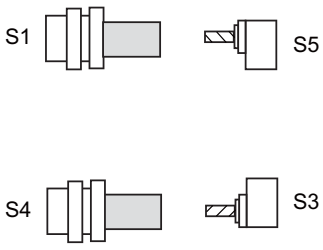
- Spindle synchronization I
- Tool spindle synchronization IA

Note

(1) The guide bushing spindle synchronization is not included in the multiple spindle synchronization sets.

This function enables the following machining.

[Example for Lathe systems]

<table>
<tr>
<td>Spindle synchronization I command (S1-S2) + Tool spindle synchronization IA command (S3-S4)

</td>
<td>Spindle synchronization I command (S1-S2) + Tool spindle synchronization IA command (S1-S5)

</td>
</tr>
<tr>
<td>Spindle synchronization I command (S1-S2) + Tool spindle synchronization II command (S3-C4)

</td>
<td>Spindle/C axis synchronization control command (S1/C1-S2) + Tool spindle synchronization II command (S5-C1)

</td>
</tr>
<tr>
<td>Spindle synchronization I command (S1-S2) + Spindle superimposition control command (S4-S3)

</td>
<td>Tool spindle synchronization IA command (S5-S1) + Tool spindle synchronization IA command (S3-S4)

</td>
</tr>
<tr>
<td>Tool spindle synchronization II command (S5-C1) + Tool spindle synchronization II command (S3-C4)

</td>
<td>Spindle superimposition control command (S1-S5) + Spindle superimposition control command (S4-S3)

</td>
</tr>
</table>

■ Command format

For details on the command format which starts each function, refer to the respective chapters.

[Spindle synchronization control cancel]

- Command to cancel all synchronization modes
  Cancels all the synchronized sets. The format varies depending on the parameter settings.

G113 H0 ;	**Cancels all the active tool spindle synchronization II (hobbing) commands.**

G113 D0 ;	**Cancels all the active Spindle synchronization I, Tool spindle synchronization IA (Spindle-Spindle, Polygon), and Spindle superimposition commands.**

G113 ;

- Spindle synchronization I, Tool spindle synchronization IA (Spindle-Spindle, Polygon), or Spindle superimposition control cancel command

G113 D_ ;

D	Synchronized or superimposed spindle to cancel (For a spindle number command, the range is from 1 to the number of available spindles, or from 1 to 9 for a spindle name command.)

- Tool spindle synchronization II (hobbing) cancel command (when the G code list of the program is 2, 3, 4, or 5)

G113 H_ ;

H	Reference spindle to cancel (For a spindle number command, the range is from 1 to the number of available spindles, or from 1 to 9 for a spindle name command.)

- Tool spindle synchronization II (hobbing) cancel command (when the G code list of the program is 6 or 7)
  Cancel tool spindle synchronization II (hobbing) of the specified part system.

G80.4 ;

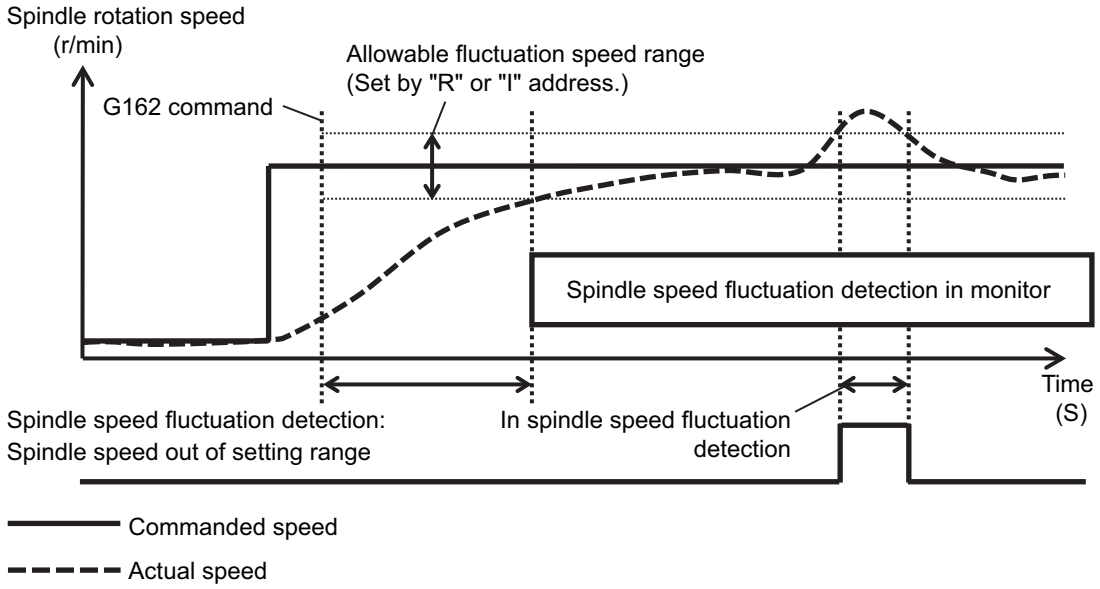
- Tool spindle synchronization IB (spindle-spindle, polygon) cancel command
  Cancel tool spindle synchronization IB (spindle-spindle, polygon) of the specified part system.

G50.2 ;

8.1.16 Spindle Speed Fluctuation Detection

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

When this function is valid and the spindle actual speed fluctuates for the commanded speed by the program due to external factors such as load fluctuation, the NC outputs the signal (Spindle speed out of setting range) to PLC and the operation error occurs. PLC can take the necessary measure for the fluctuation of the spindle speed using the output signal (Spindle speed out of setting range) from the NC. The operation error output from the NC does not stop the automatic operation or the spindle. A parameter is used to switch whether or not to output the operation error during spindle speed fluctuation detection (G162).



■ Command format

Starting the spindle speed fluctuation detection
The spindle speed fluctuation detection is started by commanding G162.

G162 S__ P__ Q__ R__ I__ ;	
S	: Detection target spindle name (1 to 9)
P	: Delay time for starting spindle speed fluctuation detection (0 to 99.999 (s))
Q	: Spindle speed attainment detection range (1 to 100 (%))
R	: Spindle speed fluctuation allowance rate (1 to 100 (%))
I	: Allowable fluctuation range in spindle speed (0 to 999999 (r/min))

Canceling the spindle speed fluctuation detection
The spindle speed fluctuation detection is canceled by commanding G163.

G163 S__ ;	
S	: Detection target spindle name (1 to 9)

8.1.17 Spindle Motor Temperature Output to PLC

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

This function outputs spindle motor temperature information to R registers.
Spindle motor temperature can be monitored by this function. The function makes it possible, for instance, to add MTB-original custom spindle control based on the spindle motor temperature, and to notify external equipment of temperature information.

8.2 Tool Functions (T)

8.2.1 Tool Functions (T Command)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

The command is issued with an 8-digit number following address T (T0 - T99999999). The tool function is used to command the tool No. In the lathe specification controller, the tool compensation (tool length compensation, tool nose wear compensation) Nos. are also indicated.

[M system]
Tool function, or T function, is used to designate the tool No. and tool compensation No.
This can be designated with an 8-digit number following address T (0 to 99999999). Up to four sets of T commands can be issued in one block. However, the number of M commands that can be issued within the same block depends on the parameter setting.
BCD output or binary output can be selected by parameter.
Output signal is 8-digit BCD code and start signal, signed 32-bit binary data and start signal, or non-signed 32-bit binary data and start signal.
T function can be designated with any other kind of commands. In the case where a movement command is in the same block, two different command sequences are available. Depending on machine specifications, either one of the following two will be applied.

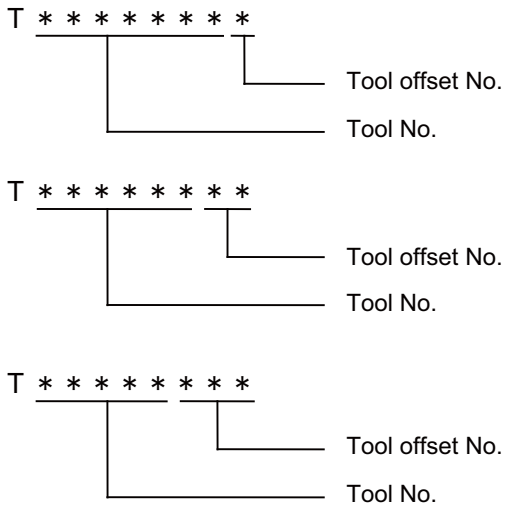
- T function is executed after the movement is completed.
- T function is executed at the same time as when the movement command is issued.

Processing and completion sequences are required for all the T commands.

Note

(1) In some setting and display units, there may be screens that cannot display all eight digits.

[L system]
The command is issued with an 8-digit number following address T (T0 to T99999999). The high-order digits are designated as the tool No., and the low-order digit(s) are designated as the offset No. Each number of high/low-order digits depends on the specifications of MTB (the parameter setting).



The tool No. code data and start signal will be output to the PLC.
All the other details are the same as in M system.

8.3 Miscellaneous Functions (M)

8.3.1 Miscellaneous Functions

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

Miscellaneous function, or M function, is used to command auxiliary functions for NC, such as rotating the spindle forward/ backward or stopping it, as well as turning the cooling oil ON/OFF.
These functions are designated by an 8-digit number (0 to 99999999) following the address M, and multiple commands can be issued in a single block. The number of M commands that can be issued within the same block depends on the parameter setting. (Maximum 4 sets)
BCD output or binary output can be selected by parameter.
Output signal is 8-digit BCD code and start signal, signed 32-bit binary data and start signal, or non-signed 32-bit binary data and start signal.

(Example) G00 Xx1 Mm1 Mm2 Mm3 Mm4 ;

- When the number of M commands in a single block is greater than the setting value of the parameter, the commands issued later are valid.
  M00, M01, M02, M30, M96, M97, M98 and M99 are used only for some specific purposes and cannot be assigned as regular M functions.
- Processing and completion sequences are required for all M functions except M96, M97, M98 and M99.
  Refer to user PLC specification for the relationship between values and functions.
  As for M00, M01, M02 and M30, next block is not read into pre-reading buffer due to ban on pre-reading processing.
- Although M00, M01, M02, and M30 output an independent signal for each, the independent output of M00, M01, M02 and M30 will be reset upon pressing a reset key.
- M command can be designated with other commands within the same block. In the case where a movement command is in the same block, two different command sequences are available. Depending on user PLC specifications, either one of the following two will be applied.
  - M function is executed after the movement is completed.
  - M function is executed at the same time as when the movement command is issued.

  Processing and completion sequences are required for all the M commands except M96, M97, M98 and M99.

Note

(1) In some setting and display units, there may be screens that cannot display all eight digits.

8.3.2 Multiple M Codes in 1 Block

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

Multiple sets of M commands can be issued in a block. (Maximum 4 sets)
Respective processing and completion sequences are required for all M commands included in a block. (except M96, M97, M98 and M99.)

Note

(1) This function requires a built-in PLC. In this case, the code data and start signals of all the M commands in the same block are transferred simultaneously from the controller to the PLC, and so high-speed machine control can be done by the PLC processing sequence.

8.3.3 M Code Independent Output

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

When the M00, M01, M02 or M30 command is assigned during an automatic operation (tape, memory, MDI) or by a manual numerical command, the signal of this function is output. It is turned OFF after the miscellaneous function finishes or by the "Reset & rewind" signal.

Machining program	M code independent output	Response to controller
M00	M00	Fin1 or Fin2
M01	M01	Fin1 or Fin2
M02	M02	Reset & rewind
M30	M30	Reset & rewind

If movement or dwell command exists in the same block as these M commands, this signal is output upon completion of the movement or dwell command.

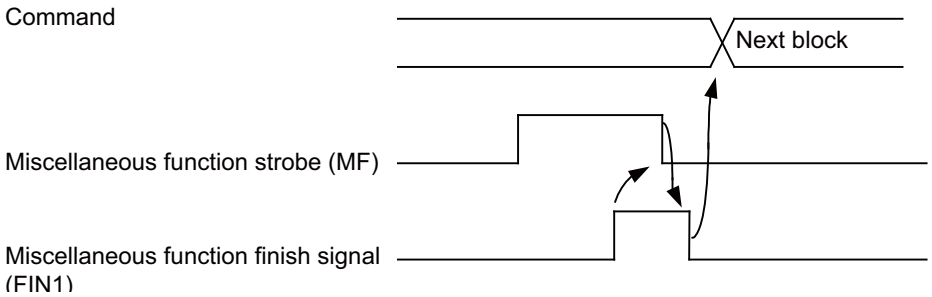
8.3.4 Miscellaneous Function Finish

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

These signals inform the CNC system that a miscellaneous function (M), spindle function (S), tool function (T) or 2nd miscellaneous function (A, B, C) has been assigned and that the PLC which has received it has completed the required operation. They include miscellaneous function finish signal 1 (FIN1) and miscellaneous function finish signal 2 (FIN2).

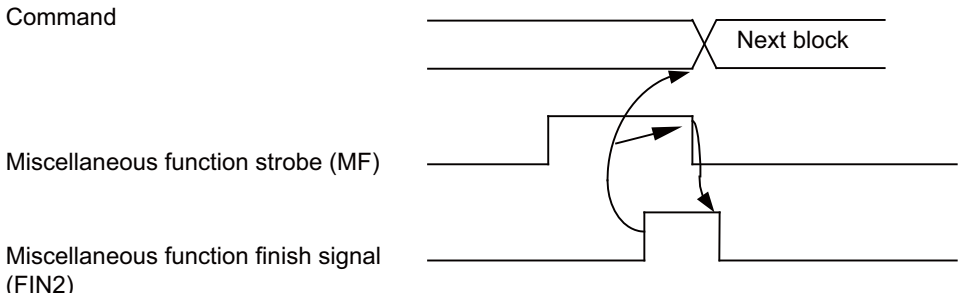
■ Miscellaneous function finish signal 1 (FIN1)

When the controller checks that FIN1 is ON, it sets the function strobes OFF. Furthermore, when the PLC checks that the function strobes are OFF, it sets FIN1 OFF. The controller checks that FIN1 is OFF and advances to the next block.
Below is an example of a time chart applying when a miscellaneous function has been assigned.



■ Miscellaneous function finish signal 2 (FIN2)

When the controller checks that FIN2 is ON, it sets the function strobes OFF and simultaneously advances to the next block. The PLC checks that the strobe signals are OFF and sets FIN2 OFF.
Below is an example of a time chart applying when a miscellaneous function has been assigned.



8.3.5 M Code Output during Axis Traveling

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	O	O	O	O	O	O	—

This function controls the timing at which miscellaneous functions are output. It outputs a miscellaneous function when a traveling axis reaches a designated position.

The command format is as follows.

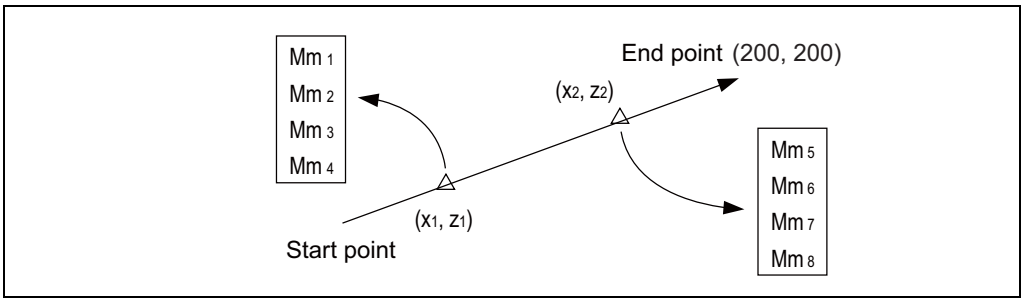
G117 Xx1 Zz1 Cc1 □□□□ ;

G117	: Command of M code output during axis traveling
Xx1,Zz1,Cc1	: Movement start points
□□□□	: Miscellaneous function

- This command is issued independently immediately before the block with the movement command that activates the miscellaneous function.
- Single block stop does not apply to this command.
- The maximum number of groups to which the miscellaneous functions in the G117 block can be issued is as follows:

M commands	4 sets
S commands	2 sets
T commands	1 set
2nd miscellaneous function	1 set

- This command can be issued in up to two consecutive blocks.
  When issued in three or more consecutive blocks, the last two blocks will be valid.
  (Example)
  G117 Xx_1 Zz_1 Mm_1 Mm_2 Mm_3 Mm_4 ;
  G117 Xx_2 Zz_2 Mm_5 Mm_6 Mm_7 Mm_8 ;
  G01 X200 Z200 ;
  :



8.3.6 Miscellaneous Function Command High-speed Output

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

The miscellaneous command high-speed output shortens a processing time per miscellaneous function. There are two functions that can be selected individually.

■ Change miscellaneous command completion method

Select one of the following methods by the parameter.

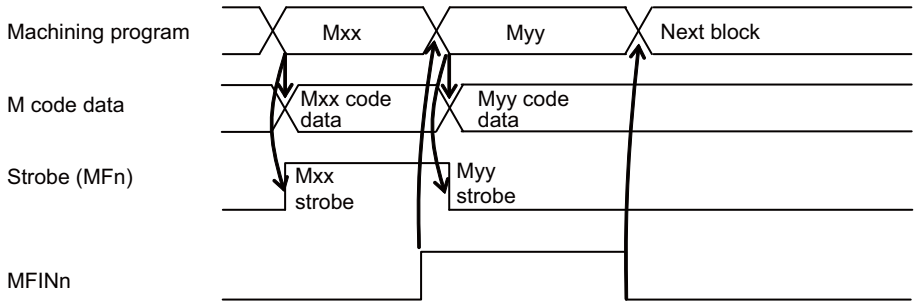
- High-speed method:
  The controller inverts a strobe signal logically at the time of outputting a miscellaneous command. The PLC performs the designated operation and logically inverts the "High-speed miscellaneous function finish" signal. The controller completes the miscellaneous function when the strobe signal and the "High-speed miscellaneous function finish" signal become the same logic level.
- Normal method:
  Conventional method (Refer to " 8.3.4 Miscellaneous Function Finish" for details.)

■ Selecting miscellaneous command completion method

Select whether or not to wait for a finish signal from the PLC using the parameter.
It can be selected for M, S, T or B individually.

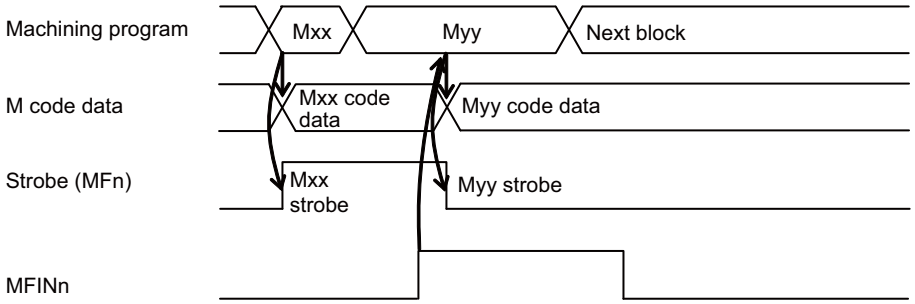
High-speed method that waits for a finish signal from the PLC

The miscellaneous function completes by matching the logic level of the finish signal with that of the strobe signal.



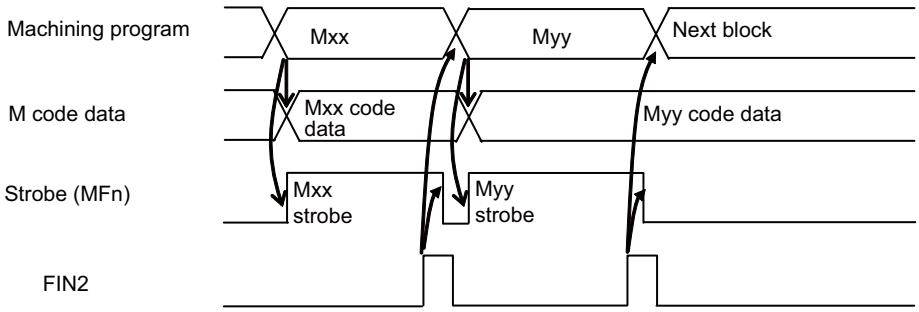
High-speed method that does not wait for a finish signal from the PLC

When a series of miscellaneous commands is issued, the completion of the previous miscellaneous command is waited.



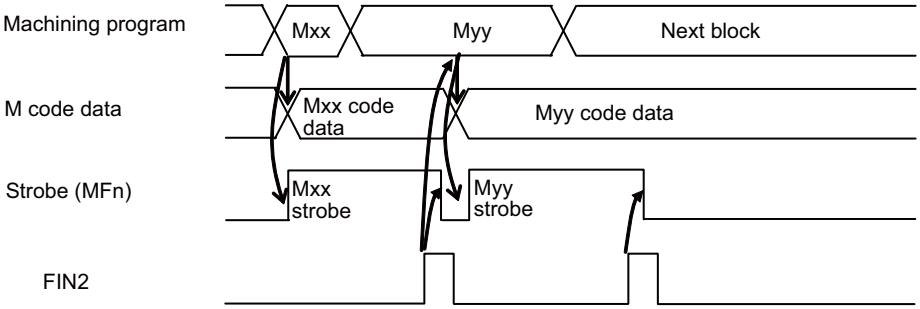
Normal method that waits for a finish signal from the PLC

Conventional method.



Normal method that does not wait for a finish signal from the PLC

When a series of miscellaneous commands is issued, the completion of the previous miscellaneous command is waited.



8.4 2nd Miscellaneous Functions (B)

8.4.1 2nd Miscellaneous Functions

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

The code data and start signals are output when an 8-digit number is assigned following the address code A, B or C - whichever does not duplicate the axis name being used.
Processing and complete sequences must be incorporated on the PLC side for all 2nd miscellaneous commands.

Note

(1) This function requires a built-in PLC.
(2) There are some screens in the setting and display unit that cannot display all eight digits.

8.4.2 2nd Miscellaneous Function Name Extension

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

The 2nd miscellaneous function name same as the additional axes (A, B, C) can be used by specifying the command address of the 2nd miscellaneous function with two characters.
Whether the command address has one character or two characters can be set by the parameter.
The operation of the 2nd miscellaneous function is same as the operation with the one character address method.

9

Tool Compensation

9.1 Tool Length/Tool Position

9.1.1 Tool Length Offset

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

These commands make it possible to control the axis movement by compensating the position of the end point of the movement command by a compensation amount set on the tool compensation screen.
Using this function, it is possible to compensate the difference in distance between the actual position of the machine's tool nose and the program coordinate position made by the tool length and to enhance both the programming and operational efficiency.

[M system]

```
G43 Zz1 Hh1 ;
```

```
G44 Zz1 Hh1 ;
```

G43	Tool length compensation command + direction (z1+h1)
G44	Tool length compensation command - direction (z1-h1)
Zz1	Compensation axis Tool length compensation can be provided not only for the Z axis but for all the other axes (X, Y, etc.) which can be controlled in the system.
Hh1	Compensation No.

Compensation can be canceled by the following G commands.

```
G49 ;
G43 H0 ;
G44 H0 ;
```

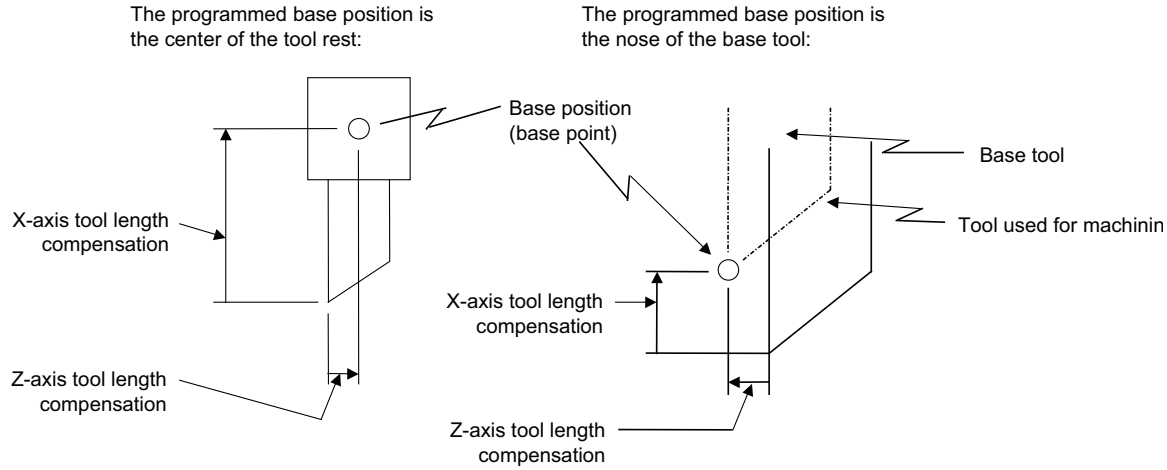
Note

(1) When the tool length compensation axis is returned to the reference position, the compensation of that axis is canceled.

[L system]

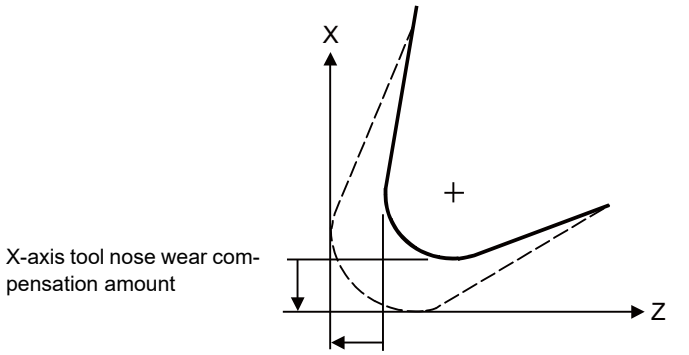
■ Shape compensation

Tool length is compensated in reference to the programmed base position. Generally, the center of the tool rest or the tip of the base tool is used as the programmed base position.



■ Wear compensation

The wear of a tool nose can be compensated.

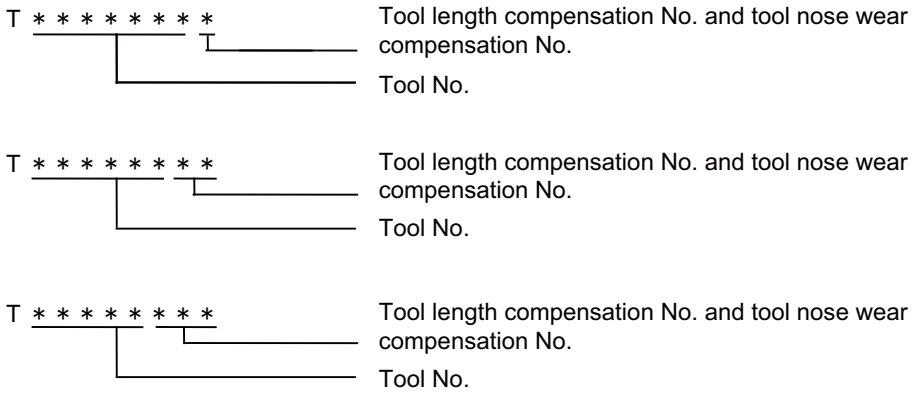


Z-axis tool nose wear compensation amount

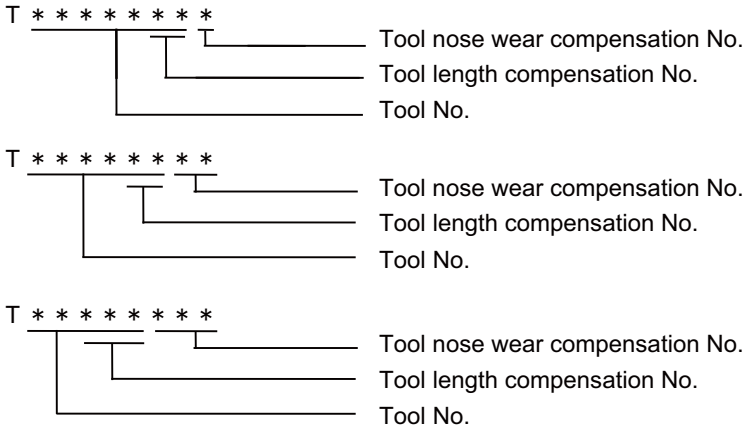
■ Command format

Tool compensation is performed by a T command. It is specified in eight digits following address T. Tool compensation is divided into two types: tool length compensation and tool nose wear compensation. It depends on the machine specifications as to which digit of T command denotes the tool No., tool length compensation No. or tool nose wear compensation No. (specified by parameters).

[Specifying tool length and wear compensation Nos. together using low-order digits of the T command]



[Specifying tool length and wear compensation Nos. separately]



The tool compensation for the lathe is valid only for the X and Z axes. If an additional axis (Y axis) and the second additional axis are added, the tool compensation will be validated for each additional axis. It supports 4 axes in total.
The additional axis is the third or fourth axis which is selected using a parameter.

9.1.2 Tool Position Offset

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	—	—	—	—	—	—	—

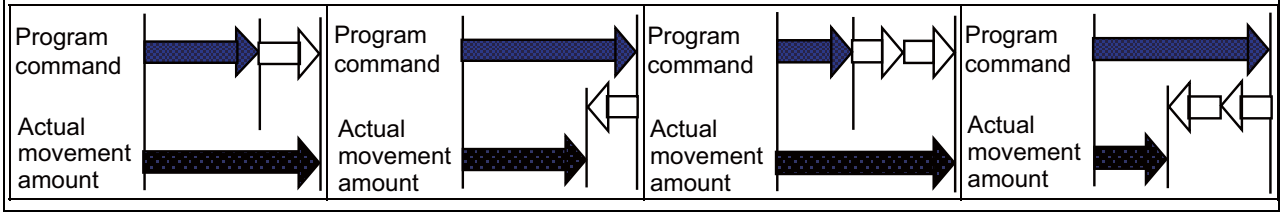
This function uses commands to control the movement by changing the positions of the end points of the movement commands to positions which have been extended or reduced by an amount equivalent to the tool compensation amount. This function can be used to compensate for the difference in distance between the position where the tool on the machine is actually mounted and the programmed coordinate position based on the tool position and thereby improve the efficiency of both machining and operation.

G45 G00 Xx1 Yy1 Dd1 ;

G45	: Tool position offset command
Xx1,Yy1	: Movement axis
Dd1	: Offset No.

With tool position offset, the offset operation is performed only for blocks containing a G45 to G48 command.

G45 command	G46 command	G47 command	G48 command
Extension by amount equivalent to offset amount	Reduction by amount equivalent to offset amount	Extension by twice the offset amount	Reduction by twice the offset amount



- If the start and end points are on an axis, the radius can be extended or reduced only for one-quarter, one-half and three-quarter arcs.
- In the case of absolute commands, the position is extended or reduced in each axial direction from the end point of the previous block along the line of the movement toward the position commanded in the block containing the G45 (or G46, G47 or G48) command.
- In the case of simultaneous n axes command, the same amount of offset is applied to all the axes that have the command within the range of the number of simultaneous contouring control axes.
  Tool position offset is also valid for additional axes.

9.1.3 Tool Compensation for Additional Axes

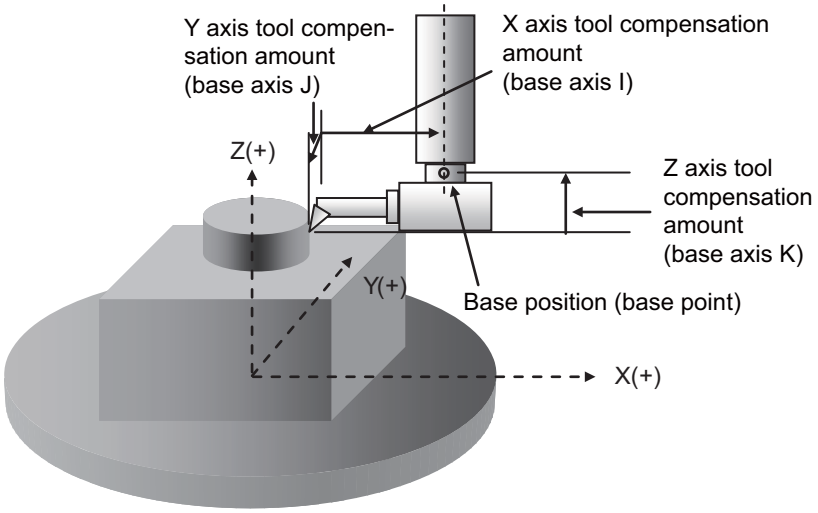
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	O	O	O	O	O	O	O

The tool compensation for the lathe is valid for the first axis (normally X axis) and the second axis (normally Z axis). If additional or second additional axes are added as the third and subsequent axes, the tool compensation is also valid for each additional axes.
The additional axes to which the tool compensation is executed are determined according to the parameter setting.

9.1.4 Tool Position Compensation (G43.7)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	—	—	—
L	—	—	—	—	—	—	—

The position compensation of a turning tool is executed when turning is performed in a machine of machining center system. Use of the tool position compensation enables the three base axes (X, Y and Z axes) to be compensated from the tool base position (base point).
Before setting the compensations of the three base axes, set the parameter to switch the tool compensation display type to type III.



■ Command format

G43.7 H__;	Tool position compensation start

G49;	Tool position compensation cancel

H	Compensation No. (H0 cancels tool position compensation.)

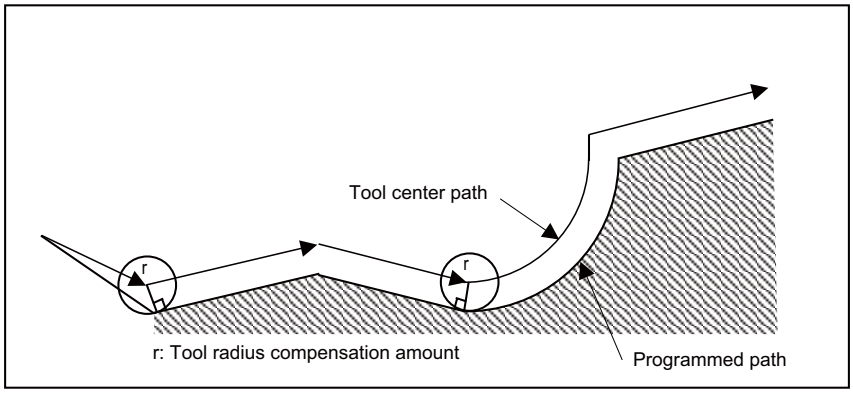
9.2 Tool Radius

9.2.1 Tool Radius Compensation

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	—	—	—	—	—	—	—

These commands function to provide tool radius compensation. Through a combination with the G command and D address assignment, they compensate for the actual tool center path either inside or outside the programmed path by an amount equivalent to the tool radius.
The tool path is calculated by the intersection point arithmetic system and, as a result, excessive cutting amounts on the inside of corners are avoided.

G code	Function
G38	Vector designation during tool radius compensation
G39	Corner arc during tool radius compensation
G40	Tool radius compensation cancel
G41	Tool radius compensation left command
G42	Tool radius compensation right command



The tool radius compensation command controls the compensation from that block in which G41 or G42 is commanded.
Whether in continuous or single block operation, compensation always starts after reading three blocks, or if the three blocks do not contain any movement command, up to five continuous blocks will be pre-read.
In compensation mode, too, up to five blocks are pre-read and the compensation is arithmetically processed.
In the compensation mode, the program is read up to five blocks ahead including blocks with no movement, and interference check using tool radius is conducted up to three blocks ahead in any of those blocks with movement.
The maximum number of blocks pre-read at start of the compensation and during the compensation mode can be changed by the parameter.

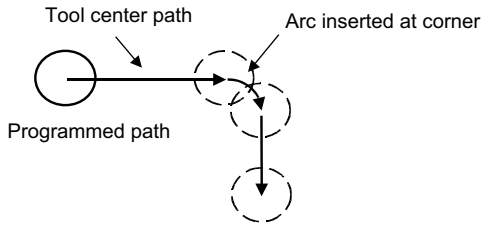
G17 G01 G41 Xx1 Yy1 Dd1 ;

G17	Compensation plane
G01	Cutting command
G41	Tool radius compensation left command
Xx1,Yy1	Movement axis
Dd1	Compensation No.

The compensation plane, movement axes and next advance direction vector are based on the plane selection command designated by G17 to G19.
G17: XY plane, X, Y, I, J
G18: ZX plane, Z, X, K, I
G19: YZ plane, Y, Z, J, K

An arc is inserted at the corner by the following command during tool radius compensation.

G39 Xx1 Yy1 ;	
G39	: Corner arc during tool radius compensation
Xx1,Yy1	: Movement amount



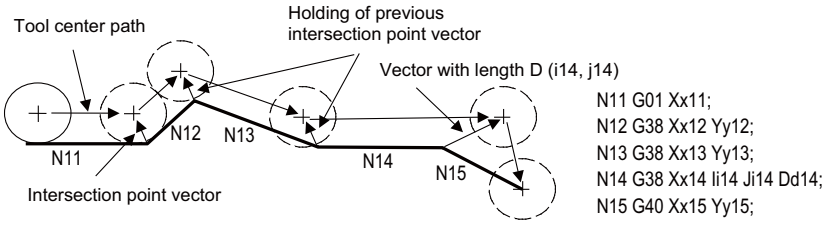
The compensation vector can be changed in following two ways.

G38 Xx1 Yy1 ;	
G38	: Vector designation for tool radius compensation
Xx1,Yy1	: Movement amount

The tool radius compensation vector amount and direction are retained.

G38 Xx1 Yy1 Ii1 Jj1 Dd1 ;	
G38	: Vector designation for tool radius compensation
Xx1,Yy1	: Movement amount
Ii1,Jj1	: Compensation vector direction
Dd1	: Compensation vector length

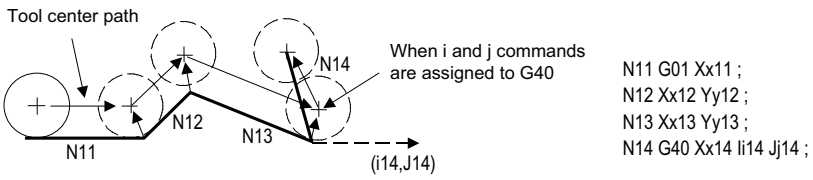
The tool radius compensation vector direction is updated by I and J.



The tool radius compensation is canceled by the following command.

G40 Xx1 Yy1 Ii1 Jj1 ;	
G40	: Tool radius compensation cancel
Xx1,Yy1	: Movement amount
Ii1,Jj1	: Compensation vector direction

The vector prior to canceling is prepared by calculating the intersection point with the I and J direction.



9.2.2 3-Dimensional Tool Radius Compensation

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	—	—	—
L	Δ(*1)	Δ(*1)	—	—	—	—	—

(*1) This function is available during program format switch.

This command serves the function of compensating for the spherical radius of ball end mills. It compensates for the actual tool center path to be either more outside or inside the programmed path by an amount equivalent to the tool radius amount in accordance with the 3-dimensional vectors.

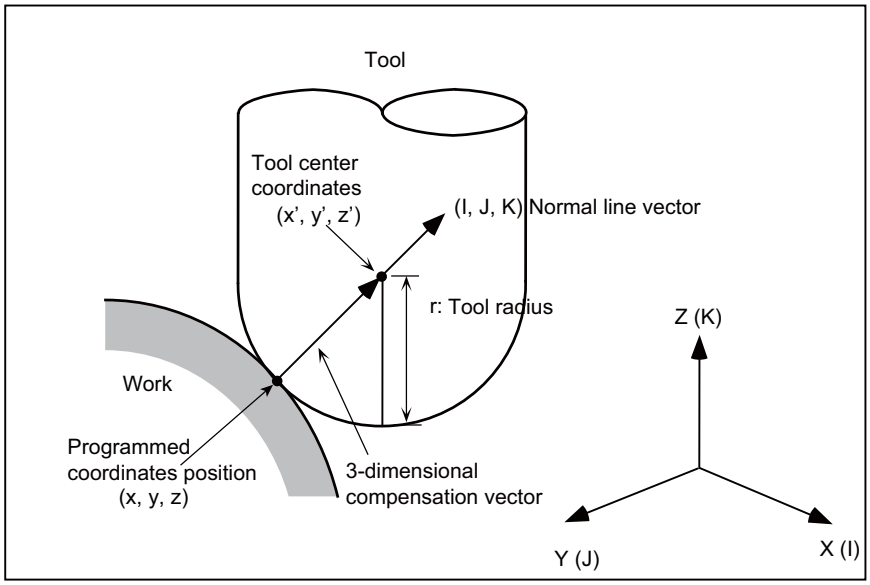
G code	Function
G40	Tool radius compensation cancel
G41	Tool radius compensation left command (compensation in the direction toward IJK)
G42	Tool radius compensation right command (compensation in the direction away from IJK)

With 3-dimensional tool radius compensation, the compensation is controlled from the block in which the block containing the G41 or G42 command has been designated.

G01 G41 Xx1 Yy1 Zz1 Ii1 Jj1 Kk1 Dd1 ;

G01	: Cutting command
G41	: Tool radius compensation left command
Xx1,Yy1,Zz1	: Movement axis
Ii1,Jj1,Kk1	: Tool sphere center vectors
Dd1	: Compensation No.

The radius from the tool spherical center to the end nose serves as the compensation amount that corresponds to the tool compensation No. designated by d1.
The compensation Nos. that can be used are limited by the "number of tool compensation sets." (Refer to " 9.3.1 Number of Tool Offset Sets".)
If the compensation vectors (tool spherical center vectors) are to be changed, the G41 and I, J and K commands must be assigned.

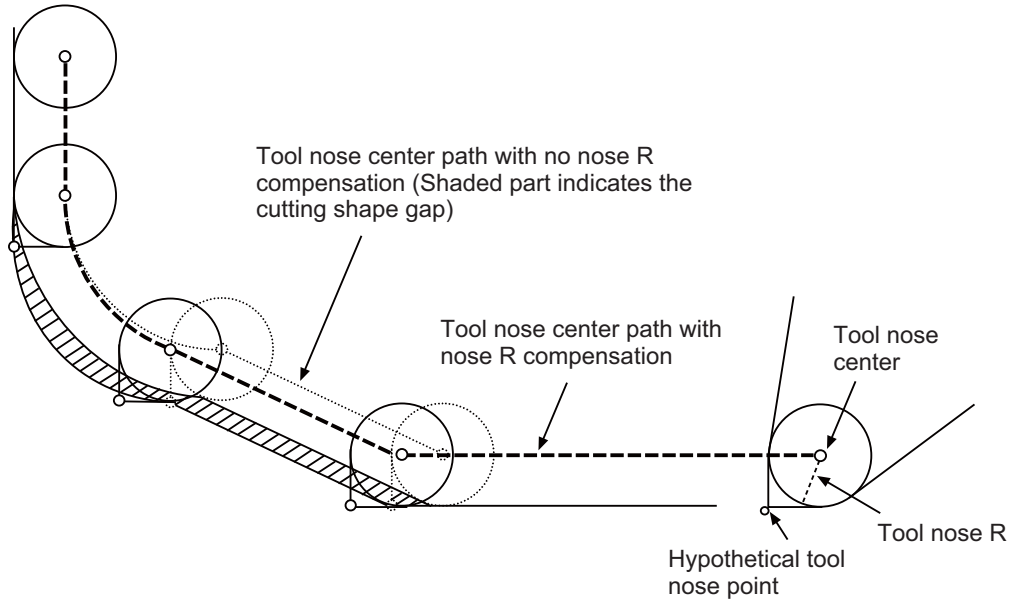


9.2.3 Tool Nose Radius Compensation (G40/41/42)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	—	—	—
L	O	O	O	O	O	O	O

Because a tool nose is generally rounded, a hypothetical tool nose point is used for creating programming. As a result, due to this roundness of the tool nose, there will be a gap between the programmed shape and the actual cutting shape during taper cutting or circular cutting. Nose radius compensation is a function to automatically calculate and offset this error by setting the tool nose radius value.
Nose radius compensation is a type of compensation that is available when turning is performed on a lathe.
Nose radius compensation for machining center system is a function which enables the nose radius compensation when turning is performed on a machining center. The basic operation follows the specifications of "Tool Nose Radius Compensation".

G code	Function
G40	Nose radius compensation cancel
G41	Nose R compensation left command
G42	Nose R compensation right command



Before starting a compensation, two to five blocks are pre-read for the intersection point calculation regardless of single block operation or continuous operation. (Two blocks are pre-read if there is a movement command, and up to five blocks are pre-read if not.)
Similarly, during compensation mode, up to five blocks are pre-read for compensation calculation.
The maximum number of blocks pre-read at start of the compensation and during the compensation mode can be changed by the parameter.

■ Nose R interference check

In the nose radius compensation mode, the program is read up to five blocks ahead including blocks with no movement, and an interference check using the nose radius is conducted up to three blocks ahead in any of those blocks with movement.

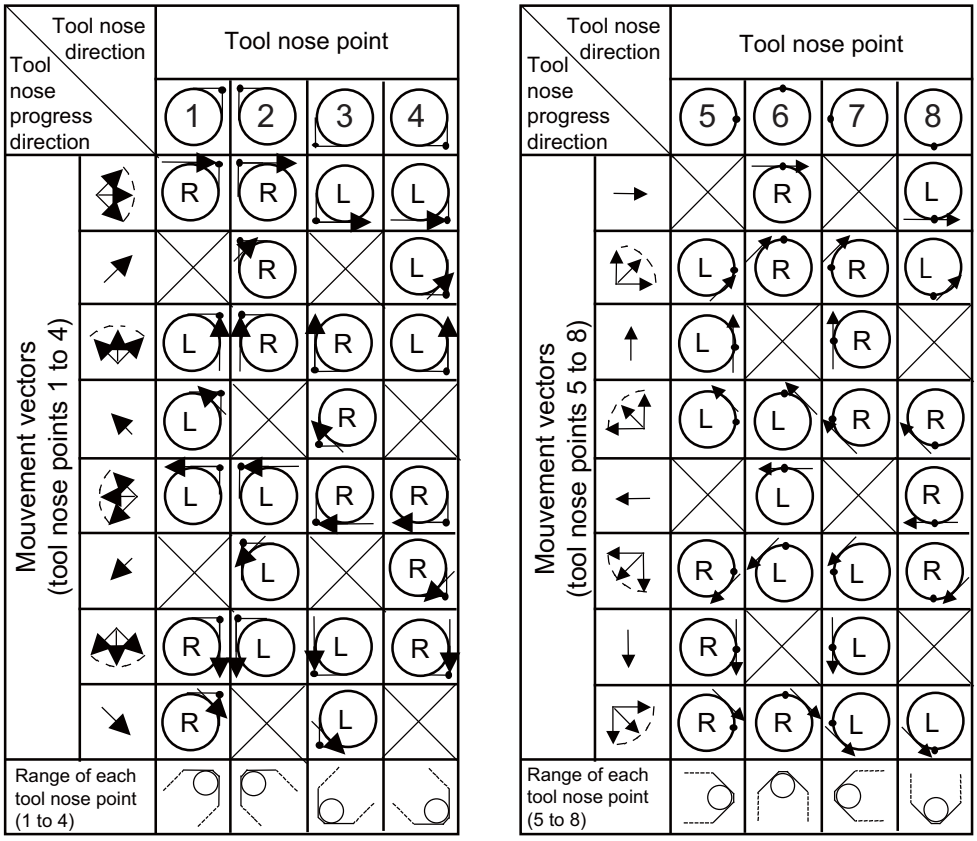
9.2.4 Automatic Decision of Nose Radius Compensation Direction (G46/40)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	O	O	O	O	O	O	O

The nose radius compensation direction is automatically determined from the tool nose point and the specified movement vector.

G code	Function
G40 G46	Nose radius compensation cancel Nose radius compensation ON (Automatic decision of compensation direction)

The compensation directions based on the movement vectors at the tool nose points are as follows:



9.2.5 Tool Radius Compensation Diameter Designation

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	—	—	—	—	—	—	—

Tool diameter designation handles the compensation amount as diameter value and compensates the amount set in the tool compensation amount screen when tool radius compensation (G41/G42 command) is commanded.
Whether compensation amount is handled in radius value or diameter value is switched by the parameter.

9.3 Tool Offset Amount

9.3.1 Number of Tool Offset Sets

[M system]

Number of tool compensation sets	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
Number of tool offset sets (99 sets)	—	—	—	—	—	—	—
Number of tool offset sets (128 sets)	—	—	—	—	—	—	—
Number of tool offset sets (200 sets)	○	○	○	○	—	—	—
Number of tool offset sets (256 sets)	—	—	—	—	—	—	—
Number of tool offset sets (400 sets)	Δ	Δ	Δ	Δ	○	○	○
Number of tool offset sets (999 sets)	Δ	Δ	Δ	Δ	—	—	—

[L system]

Number of tool compensation sets	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
Number of tool offset sets (99 sets)	—	—	—	—	—	—	○
Number of tool offset sets (128 sets)	○	○	○	○	—	—	—
Number of tool offset sets (200 sets)	—	—	—	—	—	—	—
Number of tool offset sets (256 sets)	—	—	—	—	○	○	—
Number of tool offset sets (400 sets)	Δ	Δ	Δ	Δ	—	—	—
Number of tool offset sets (999 sets)	Δ	Δ	Δ	Δ	—	—	—

The number of tool compensation sets (the number of tool offset sets) for the fixed allocation is shown in the following tables. For details on the arbitrary allocation, refer to " 9.3.3 Number of Tool Offset Sets Allocation to Part Systems".

[M system]

Function name	Common for part systems or 1-part system specification	Independent to each part system and multi-part system specification
Number of tool offset sets (200 sets)	200 sets	Divide the number of tool offset sets by the number to obtain the number of sets per part system. (If there is a remainder, the remainder is allocated to the 1st part system.)
Number of tool offset sets (400 sets)	400 sets	
Number of tool offset sets (999 sets)	999 sets	

[L system]

Function name	Common for part systems or 1-part system specification	Independent to each part system and multi-part system specification
Number of tool offset sets (99 sets)	99 sets	Divide the number of tool offset sets by the number to obtain the number of sets per part system. (If there is a remainder, the remainder is allocated to the 1st part system.)
Number of tool offset sets (128 sets)	128 sets	
Number of tool offset sets (256 sets)	256 sets	
Number of tool offset sets (400 sets)	400 sets	
Number of tool offset sets (999 sets)	999 sets	

Note

(1) Whether the tool compensation memory is provided commonly to the part systems or independently for the part systems depends on the parameter settings.

9.3.2 Offset Memory

9.3.2.1 Tool Shape/Wear Offset Amount

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

This function registers the tool shape compensation and wear compensation amounts. Compensation may encompass two or more axes.
Tool compensation types I, II and III are available for M system, which can be selected by the parameter. For L system, tool compensation type III is always used.

Shape compensation amount

The tool length compensation amount, tool radius compensation amount, nose radius compensation amount, nose radius imaginary tool tip point or tool width can be set as the shape compensation amount.
The compensation amount that can be set and used differs depending on whether compensation amount setting type I,II or III is used.

Wear compensation amount

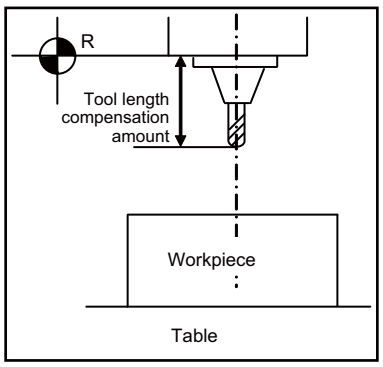
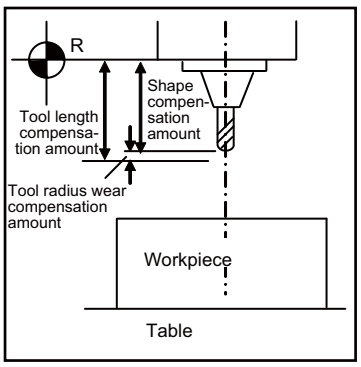
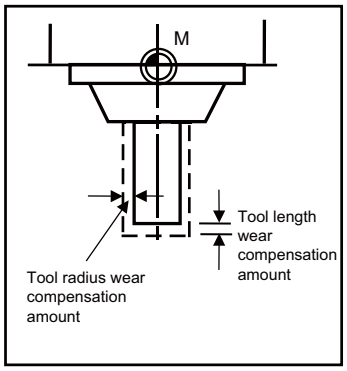
When the tip of the tool used has become worn, the wear compensation amount is used to compensate this wear. Types of wear compensation amounts include the tool length wear compensation amount, tool radius wear compensation amount, and nose radius wear compensation amount.
The wear compensation amount can be used with compensation amount setting types II and III, and it is added to the shape compensation amount for compensation.

■ Type I: 1-axis compensation amount [M system]

This is the value that is used by rotary tools.
As the tool length compensation amount, among the compensation amounts for the position of the tool moving in the direction parallel to the control axis, the compensation amount in the longitudinal direction of the rotary tool is registered. The tool length compensation amount is set as a minus value.
As the tool radius compensation amount, among the compensation amounts for the position of the tool moving in the direction parallel to the control axis, the compensation amount in the radial direction of the rotary tool is registered. The tool radius compensation amount is set as a plus value.
One compensation amount data is registered in one compensation No., and the compensation Nos. are assigned using the address D or H commands. When a No. is assigned by a D address command, compensation is provided in the form of the tool radius; when it is assigned by an H address command, it is provided in the form of the tool length.

■ Type II: 1-axis compensation amounts/with wear compensation [M system]

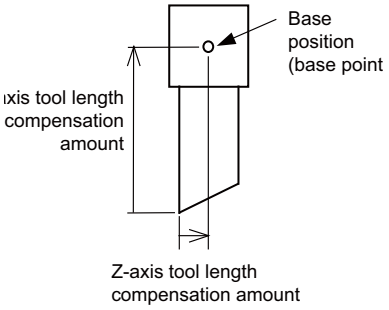
As with type I, type II is for the compensation amounts used by rotary tools.
With type II, four kinds of compensation amount data are registered in one compensation No.: the tool length compensation amount, tool length wear compensation amount, tool radius compensation amount, and tool radius wear compensation amount.
When a compensation No. is assigned by address D as the compensation amount, the tool radius is compensated using the amount obtained by adding the shape compensation amount and tool radius wear compensation amount. Further, the tool length is compensation using the amount obtained by adding the shape compensation amount and tool length wear compensation amount.

<table>
<tr><td colspan="2">Tool length compensation amount</td><td rowspan="2">Wear compensation amount when using type II</td></tr>
<tr><td>Compensation type I</td><td>Compensation type II</td></tr>
<tr>
<td>
</td>
<td>
</td>
<td>
</td>
</tr>
</table>

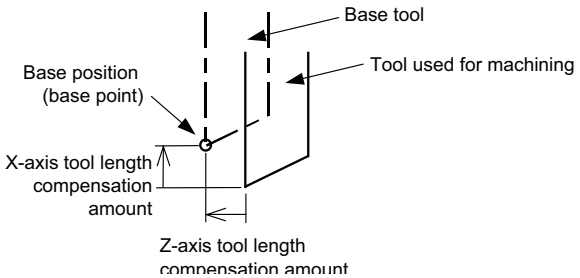
■ Type III: 2-axis compensation amounts [M system] [L system]

Type III is for the compensation amounts used by non-rotary tools.
As the compensation amounts, the tool length along the X, Y and Z axes and the wear amount along each of these axes, the nose radius and nose radius wear amount, tool tip point P and tool width can be registered.
Compensation is carried out in the directions of the X, Y and Z axes from the base position in the program. Generally, the center of the tool rest or the tip of the base tool is used as the programmed base position.

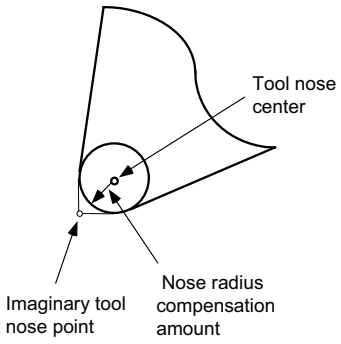
1. The programmed base position is the center of the tool rest:

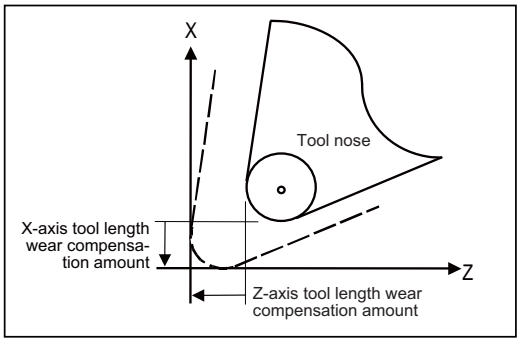


2. The programmed base position is the tip of the base tool:



The tool tip contour arc radius (nose radius) of a non-rotary tool with an arc (nose radius) at its tip is registered as the nose radius compensation amount.





The X-axis tool length compensation amount, Z-axis tool length compensation amount and nose radius compensation amount are set as plus amounts.

9.3.2.2 Compensation Type Selection by Parameter

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	—	—	—	—	—	—	—

This function switches the tool compensation type with the parameter.
The type is switched to the tool compensation type III by setting the parameter, regardless of #1037 cmdtyp.
This function enables tool compensation for a turning tool by registering the tool compensation amount of the base axes IJK and tool tip point for a machining center system.

■ Machining center system

Tool compensation type I and II are used for machining center system. However, use of this function switches the type to tool compensation type III.

[Correspondence of the registered data between the compensation types]

- Tool compensation type I -> Tool compensation type III
  The tool compensation amount of tool compensation type I is handled as tool length Z of tool compensation type III.
- Tool compensation type II -> Tool compensation type III
  The table below shows the relationship between the registered items.

Tool compensation type II	Tool compensation type III
Length	Length Z
L wear	Wear Z
Radius	Nose R
R wear	R-wear

■ Lathe system

The tool compensation type is not switched by using this function.

9.3.3 Number of Tool Offset Sets Allocation to Part Systems

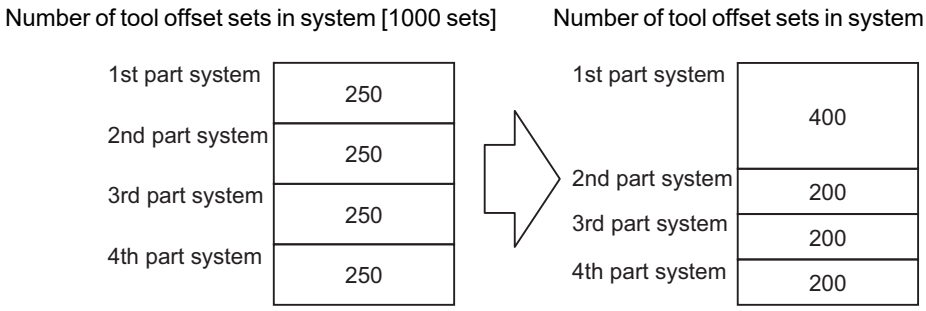
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

(*) Variable number of per-part-system tool offset sets

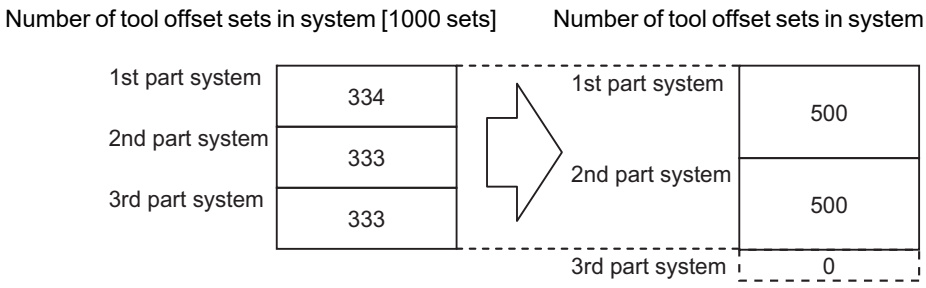
The number of tool offset sets can be set per part system.
There are two types of the allocation: "Arbitrary allocation" which allocates the number of tool offset sets to each part system arbitrarily and "Fixed allocation" which automatically allocates the number of tool offset sets to each part system equally, and the type can be selected using the parameter.
The arbitrary allocation enables the efficient allocation because when a certain part system needs only a small number of offset sets, the rest can be allocated to another part system. If an auxiliary-axis part system does not need the tool offset set at all, the number of tool offset sets can be set 0 in the auxiliary-axis part system.
Number of tool offset sets in system is the total number of tool offset sets of all part systems.

■ Arbitrary allocation

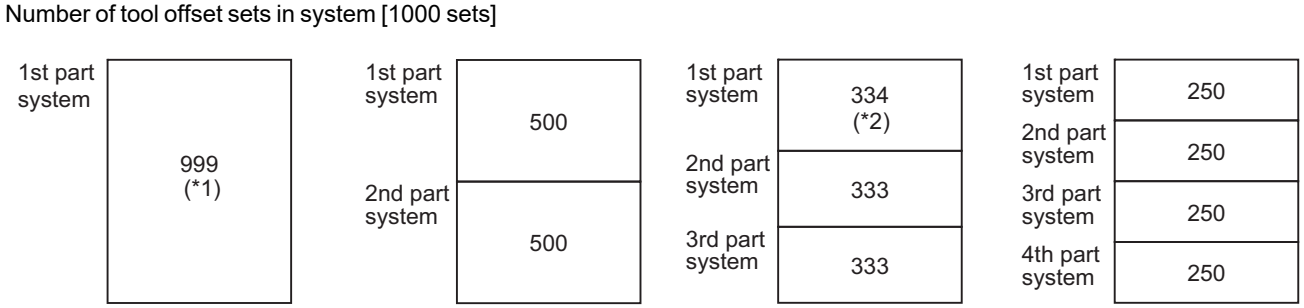
[When the number of tool offset sets is increased in the 1st part system of 4-part system]



[When the number of offset sets is set "0" to the 3rd part system of 3-part system for use of the 3rd part system as auxiliary-axis part system]



■ Fixed allocation



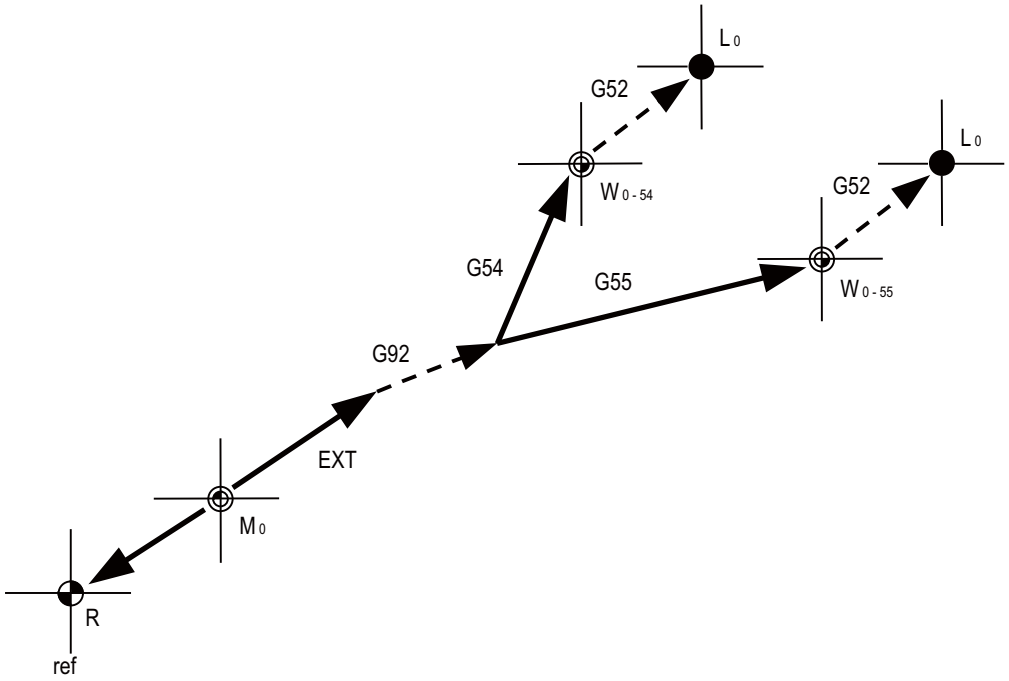
(*1) The maximum number of tool offset sets per part system is 999.
(*2) If there is any remainder, the remainder is allocated to the 1st part system.

10

Coordinate System

10.1 Coordinate System Type and Setting

The coordinate system handled by the NC is shown below.
The points that can be commanded with the movement command are points on the local coordinate system or machine coordinate system.



L0	Local coordinate system zero point	→	Offset set with parameters
G52	Local coordinate system offset(*1)	- - - →	Offset set with program
W0-54	Workpiece coordinate system zero point (G54)		(0 when power is turned ON)
W0-55	Workpiece coordinate system zero point (G55)		
G54	Workpiece coordinate system (G54) offset(*1)		
G55	Workpiece coordinate (G55) system offset		
G92	G92 coordinate system shift		
EXT	External workpiece coordinate offset		
M0	Machine coordinate system zero point		
ref	Reference position		

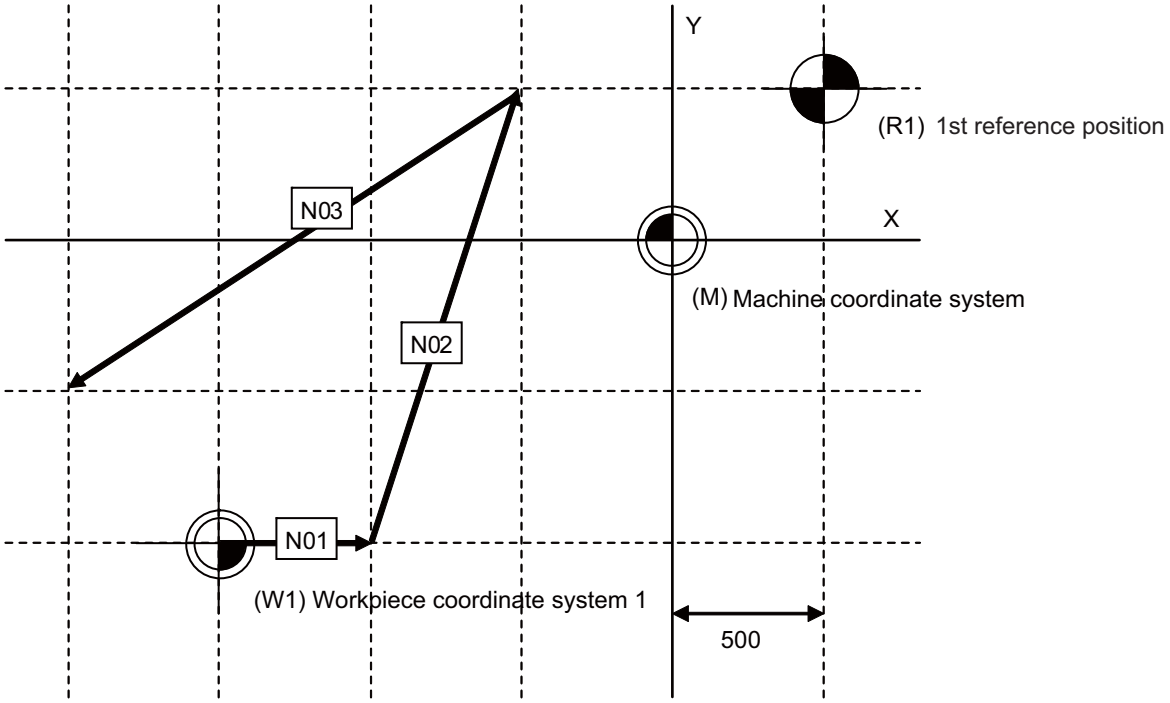
(*1) The G52 offset is available independently for G54 to G59.

10.1.1 Machine Coordinate System

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

The machine coordinate system is used to express the prescribed positions (such as the tool exchange and stroke end positions) that are specific to each machine.
The G53 command followed by coordinates moves the tool to the command position in the machine coordinate system.

If a coordinate command is included in the same block as G53, it is treated as a command not in the workpiece coordinate system but in the machine coordinate system.



When the initial position of workpiece coordinate is (0,0) with absolute command.
N01 G01 X500. Y0. F1000
N02 G53 X-500. Y500.
N03 X-500. Y500.

■ Command format

G53 X__ (U__) Y__ (V__) Z__ (W__) α__ β__ ;
X/U/Y/V/Z/W/α/β : Axis address (range of coordinate position command (mm, inch))

If the incremental or absolute commands and movement mode have been omitted, operation complies with the modal command that prevails at the time.
G53 (movement on machine coordinate system) is an non-modal command which is effective only in the block where it is assigned. The workpiece coordinate system being selected is not changed by this command.

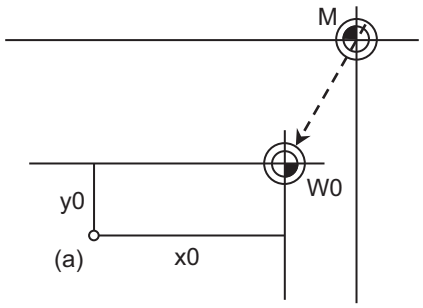
10.1.2 Coordinate System Setting

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

The zero point position of the program coordinate system can be shifted to an arbitrary position by G92 command.
As shown in the figure below, when a coordinate system setting is specified by G92, G92 shift amount is applied so that the tool position(*1) in the current workpiece coordinate system is set to the coordinate position specified by G92, and the workpiece coordinate system is shifted accordingly. All the workpiece coordinate systems (G54 to G59 and G54.1 Pn) are also shifted simultaneously at this time, but the machine does not move.

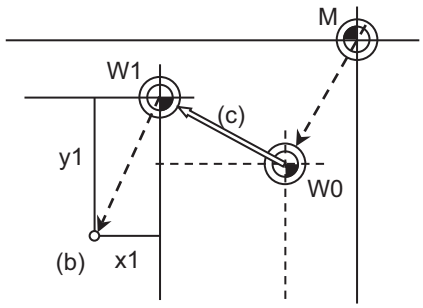
(*1) The "tool position in the current workpiece coordinate system" includes the compensation amounts for tool radius, tool length and tool position compensation.

Coordinate system created by automatic coordinate system setting



G92 Xx1 Yy1 ;

Coordinate system after coordinate system setting by G92



A workpiece coordinate system in which the current tool position is set at (x1, y1) will be newly created.

(a) Tool position
(b) G92 commanded position
(c) G92 shift amount
(W0) Workpiece coordinate system
(W1) Workpiece coordinate system after G92 command
(M) Basic machine coordinate system

The following values are set as G92 shift amounts.
X: x0 - x1
Y: y0 - y1

There are two methods to return a coordinate system shifted by G92 command to its original position.

- Carry out dog-type reference position return.
- Move the coordinate system to the machine coordinate system zero point, then command G92 and G53 in the same block to set the machine coordinate system.

G90 G53 G00 X0 Y0 ;	Positioning at machine coordinate system zero point
G92 G53 X0 Y0 ;	Coordinate system zero setting in machine coordinate system This returns all the workpiece coordinates (G54 to G59 and G54.1 Pn) to their original positions.

10.1.3 Automatic Coordinate System Setting

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

After the power is turned ON, the basic machine coordinate system and the workpiece coordinate system are automatically set without executing the zero point return.
The coordinate systems created are given below.

- Machine coordinate system corresponding to G53
- G54 to G59 workpiece coordinate system
- Local coordinate systems created under G54 to G59 workpiece coordinate systems

The distances from the zero point of G53 machine coordinate system are set to the controller coordinate related parameters.

10.1.4 Workpiece Coordinate System Selection

The workpiece coordinate systems facilitate the programming on the workpiece, serving the reference position of the machining workpiece as the zero point.
These commands enable the tool to move to the positions in the workpiece coordinate system. There are extended workpiece coordinate systems (G54.1) in addition to 6 workpiece coordinate systems, which are used by the programmer for programming (G54 to G59). The number of the extended workpiece coordinate systems varies depending on the MTB specifications.
Among the workpiece coordinate systems currently selected by these commands, any workpiece coordinate system with coordinates that have been commanded by the current position of the tool is reset. (The "current position of the tool" includes the compensation amounts for tool radius, tool length and tool position.)
A hypothetical machine coordinate system with coordinates that have been commanded by the current position of the tool is set by these commands. (The "current position of the tool" includes the compensation amounts for tool radius, tool length and tool position.) (G54,G92)

10.1.4.1 Workpiece Coordinate System Selection (6 Sets)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

Up to 6 workpiece coordinate systems can be selected.
The G54 workpiece coordinate system is selected when the power is turned ON.

G code	Function
G54	Workpiece coordinate system 1 (W1)
G55	Workpiece coordinate system 2 (W2)
G56	Workpiece coordinate system 3 (W3)
G57	Workpiece coordinate system 4 (W4)
G58	Workpiece coordinate system 5 (W5)
G59	Workpiece coordinate system 6 (W6)

■ Command format

Workpiece coordinate system selection

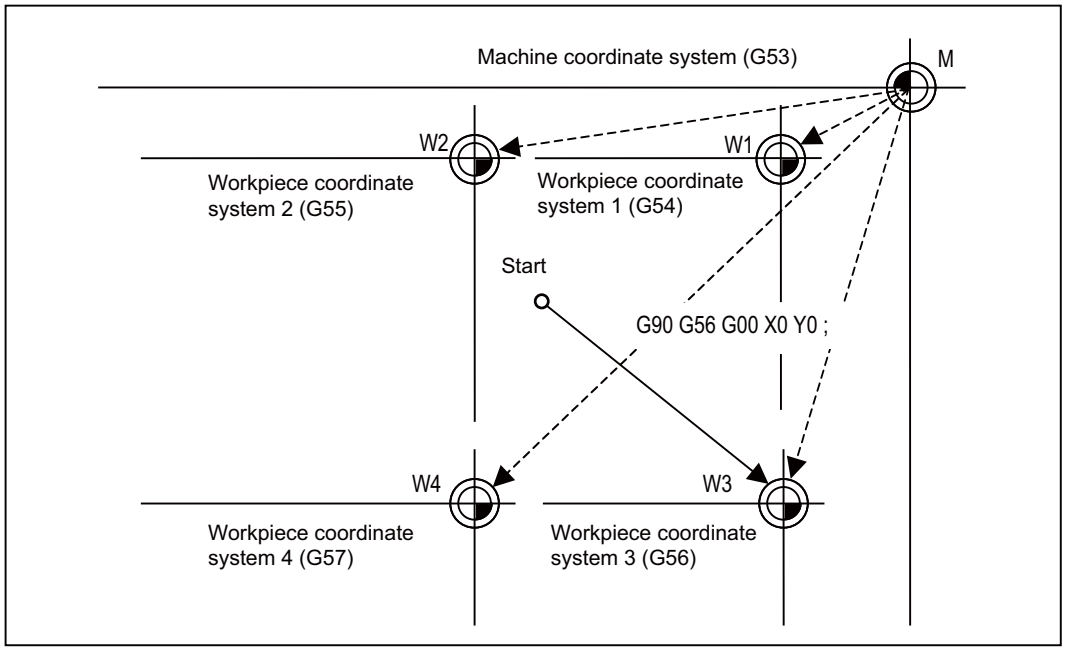
(G90) G54 to G59 ;

Workpiece coordinate system setting

(G54 to G59)G92 X__ Y__ Z__ a__ ;
X, Y, Z : Coordinate position of end point
a : Additional axis

The workpiece coordinate zero points are provided as distances from the zero point of the machine coordinate system.
Settings can be performed in one of the following two ways:

- Setting using the setting and display unit
- Setting using commands assigned from the machining program



10.1.4.2 Extended Workpiece Coordinate System Selection (48 Sets) G54.1P1 to P48

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

In addition to the six workpiece coordinate systems G54 to G59, 48 workpiece coordinate systems can be used by assigning G54.1Pn command.
The command format to select the workpiece coordinate system using the G54.1Pn command and to move on the workpiece coordinate system are given below.

Workpiece coordinate system selection

```
(G90) G54.1Pn ;
```

The numerical value n of P following G54.1 indicates each workpiece coordinate system. Specify a value between 1 and 48.
The workpiece coordinate zero points are provided as distances from the zero point of the machine coordinate system.
Settings can be performed in one of the following two ways:

- Setting using the setting and display unit
- Setting using commands assigned from the machining program

G54Pn can be used as the extended workpiece coordinate system selection command by setting the parameter.

10.1.4.3 Extended Workpiece Coordinate System Selection (96 Sets) G54.1P1 to P96

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	—	—	—
L	—	—	—	—	—	—	—

In addition to the six workpiece coordinate systems G54 to G59, 96 workpiece coordinate systems can be used by assigning G54.1Pn command.
For details, refer to " 10.1.4.2 Extended Workpiece Coordinate System Selection (48 Sets) G54.1P1 to P48".

10.1.4.4 Extended Workpiece Coordinate System Selection (300 Sets) G54.1P1 to P300

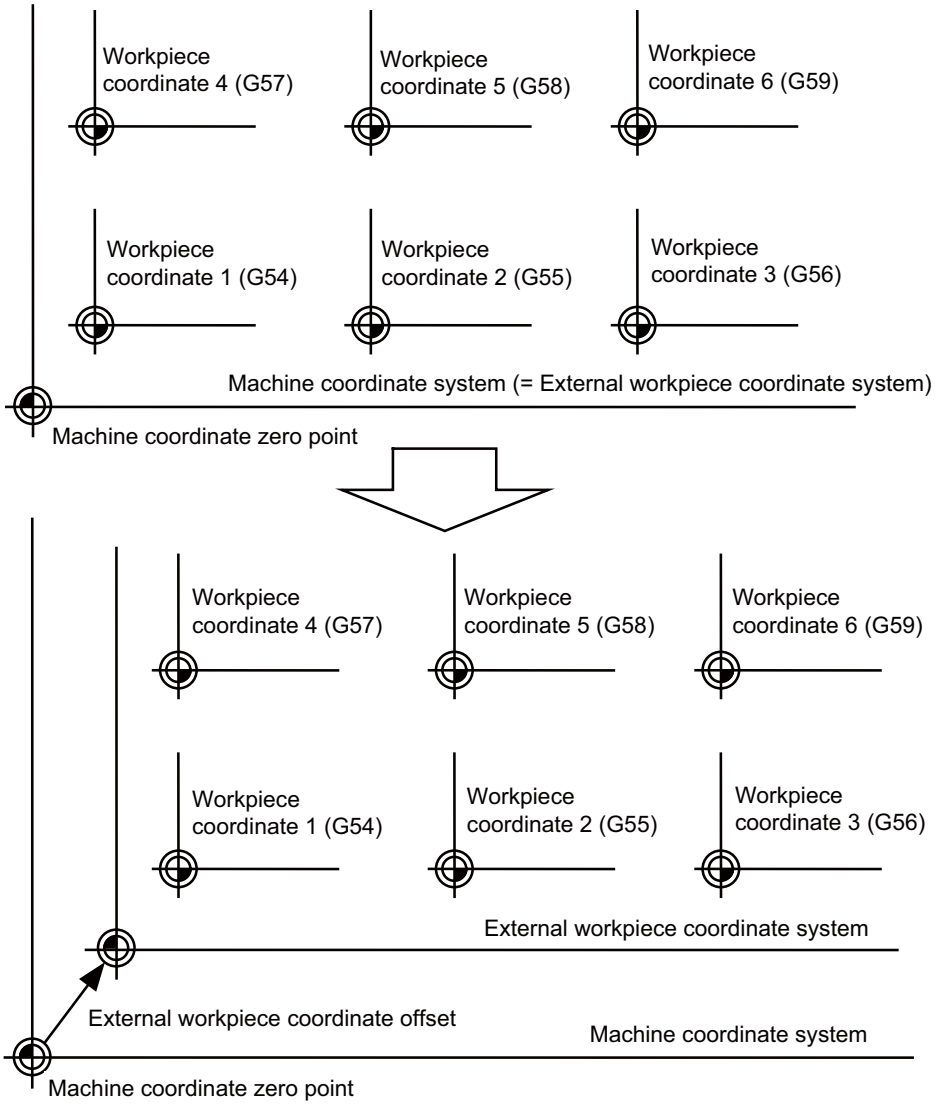
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	—	—	—
L	—	—	—	—	—	—	—

In addition to the six workpiece coordinate systems G54 to G59, 300 workpiece coordinate systems can be used by assigning G54.1Pn command. For details, refer to " 10.1.4.2 Extended Workpiece Coordinate System Selection (48 Sets) G54.1P1 to P48".

10.1.5 External Workpiece Coordinate Offset

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

External workpiece coordinate offset that serves as the reference for all the workpiece coordinate systems is available outside the workpiece coordinates.
By setting the external workpiece coordinate offset, the external workpiece coordinate system can be shifted from the machine coordinate system, and all the workpiece coordinate systems can be simultaneously shifted by an amount equivalent to the offset.
When the external workpiece coordinate offset is zero, the external workpiece coordinate systems coincide with the machine coordinate system.
It is not possible to assign movement commands with the external workpiece coordinate selected.



10.1.6 Workpiece Coordinate System Preset (G92.1)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	—	—	—
L	○	○	○	○	○	○	○

This function presets the workpiece coordinate system, which has been shifted by the programmed command or the manual operation, as the workpiece coordinate system which has been offset by the programmed command (G92.1) from the machine zero point by an amount equivalent to the workpiece coordinate offset amount.

The workpiece coordinate system is shifted from the machine coordinate system when the such operations or the programmed commands as below have been performed.

- When manual intervention has occurred in the manual absolute OFF status
- When a movement command was performed in the machine lock status
- When movement was initiated by handle interrupt
- When a movement command was performed in the mirror image mode
- When a local coordinate system was set using the G52 command
- When a workpiece coordinate system was shifted using the G92 command

Just as when manual reference position return has been performed, this function presets the workpiece coordinate system which has been shifted once to the workpiece coordinate system which has been offset from the machine zero point by an amount equivalent to the workpiece coordinate offset amount.
Furthermore, whether to preset relative coordinates as well is selected with a parameter.

■ Command format

G92.1 (G50.3) X0 Y0 Z0 α0 ; (α is the additional axis.)

Designate the addresses of the axes to be preset.
Axes whose addresses have not designated will not be preset.
Depending on the command type, G50.3 command is used in stead.
A program error results when a value other than 0 is commanded.

10.1.7 Local Coordinate System

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

The local coordinate systems can be set on the G54 through G59 workpiece coordinate systems or on the G54.1 extended workpiece coordinate system using the G52 command so that the commanded position serves as the programmed zero point. The G52 command can also be used instead of the G92 command to change the deviation between the zero point in the machining program and the machining workpiece zero point.
The local coordinate system can be designated independently among the following workpiece coordinate systems.

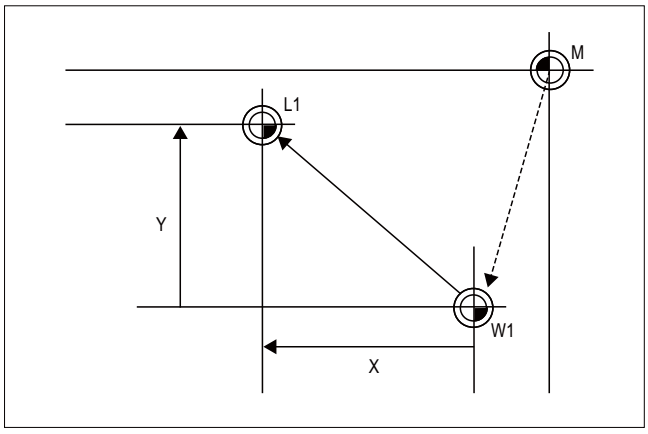
G code	Function
G54 G52	Local coordinate system on the workpiece coordinate system 1
G55 G52	Local coordinate system on the workpiece coordinate system 2
G56 G52	Local coordinate system on the workpiece coordinate system 3
G57 G52	Local coordinate system on the workpiece coordinate system 4
G58 G52	Local coordinate system on the workpiece coordinate system 5
G59 G52	Local coordinate system on the workpiece coordinate system 6
G54.1 P G52	Local coordinate system on the extended workpiece coordinate system

The command format of the local coordinate system is given below.

G54 (or G55 to G59, G54.1 P__) G52 X__ Y__ Z__ α__ ;

G54 (or G55 to G59, G54.1 P__)	Workpiece coordinate system selection, extended workpiece coordinate system selection
G52	Local coordinate system setting
X, Y, Z, α	Local coordinate system offset amount ("α" is the additional axis.)

The local coordinate zero points are provided as distances from the zero point of the designated workpiece coordinate system (local coordinate offset).
In the incremental command mode, the position obtained by adding the local coordinate offset amount to the previously specified offset amount serves as the new local coordinate zero point.
If no workpiece coordinates are designated, the local coordinates will be created on the currently selected workpiece coordinates.
This command is non-modal but the local coordinate system created by G52 is valid until the next G52 command is issued.
The local coordinate system is canceled by the input of the reset signal or by manual or automatic dog-type reference position return.



M: Machine coordinate system (G53)
W1: Workpiece coordinate system 1 (G54)
L1: Local coordinate system (G54 G52)

10.1.8 Coordinate System for Rotary Axis

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

The axis designated as the rotary axis with the parameters is controlled with the rotary axis' coordinate system.
The rotary axis includes the rotating type (short-cut valid/invalid) and linear type (workpiece coordinate position linear type, all coordinate position linear type).
The workpiece coordinate position range is 0 to 359.999° for the rotating type, and 0 to 99999.999° for the linear type.
The machine position and relative position differ according to the parameters.
The rotary axis is commanded with a degree (°) unit regardless of the inch or metric designation.
The rotary axis type can be set with the parameters for each axis.

<table>
<tr><th rowspan="3"></th><th colspan="4">Rotary axis</th><th rowspan="3">Linear axis</th></tr>
<tr><th colspan="2">Rotating type rotary axis</th><th colspan="2">Linear type rotary axis</th></tr>
<tr><th>Short-cut invalid</th><th>Short-cut valid</th><th>Workpiece coordinate position linear type</th><th>All coordinate position linear type</th></tr>
<tr><td>Workpiece coordinate position</td><td colspan="2">Displayed in the range of 0° to 359.999°.</td><td colspan="3">Displayed in the range of 0° to ±99999.999°.</td></tr>
<tr><td>Machine position/relative position</td><td colspan="3">Displayed in the range of 0° to 359.999°.</td><td colspan="2">Displayed in the range of 0° to ±99999.999°.</td></tr>
<tr><td>ABS command</td><td>The incremental amount from the end point to the current position is divided by 360, and the axis moves by the remainder amount according to the sign.</td><td>Moves with a short-cut to the end point.</td><td colspan="3">In the same manner as the normal linear axis, moves according to the sign by the amount obtained by subtracting the current position from the end point (without rounding up to 360 degrees).</td></tr>
<tr><td>INC command</td><td colspan="5">Moves in the direction of the commanded sign by the commanded incremental amount starting at the current position.</td></tr>
<tr><td rowspan="2">Reference position return</td><td colspan="5">Follows the absolute/incremental command for a movement to the interim position.</td></tr>
<tr><td colspan="3">Returns to the reference position from the interim position within a 360 degree movement.</td><td colspan="2">Moves and returns in the reference position direction for the difference from the current position to the reference position.</td></tr>
</table>

10.1.9 Plane Selection

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

G17, G18, and G19 are for specifying the planes for the arc, tool radius compensation, coordinate rotation and other such commands.

G17;	Xp-Yp plane designation
G18;	Zp-Xp plane designation
G19;	Yp-Zp plane designation

- A parameter can be used to set either the X, Y or Z axis to which the additional axis is to be parallel.
- A parameter can be used to set the initialization status (when the power has been turned ON or when the reset status has been entered) to G17, G18 or G19.
- The movement commands have no connection with the plane selection.
  (Example)

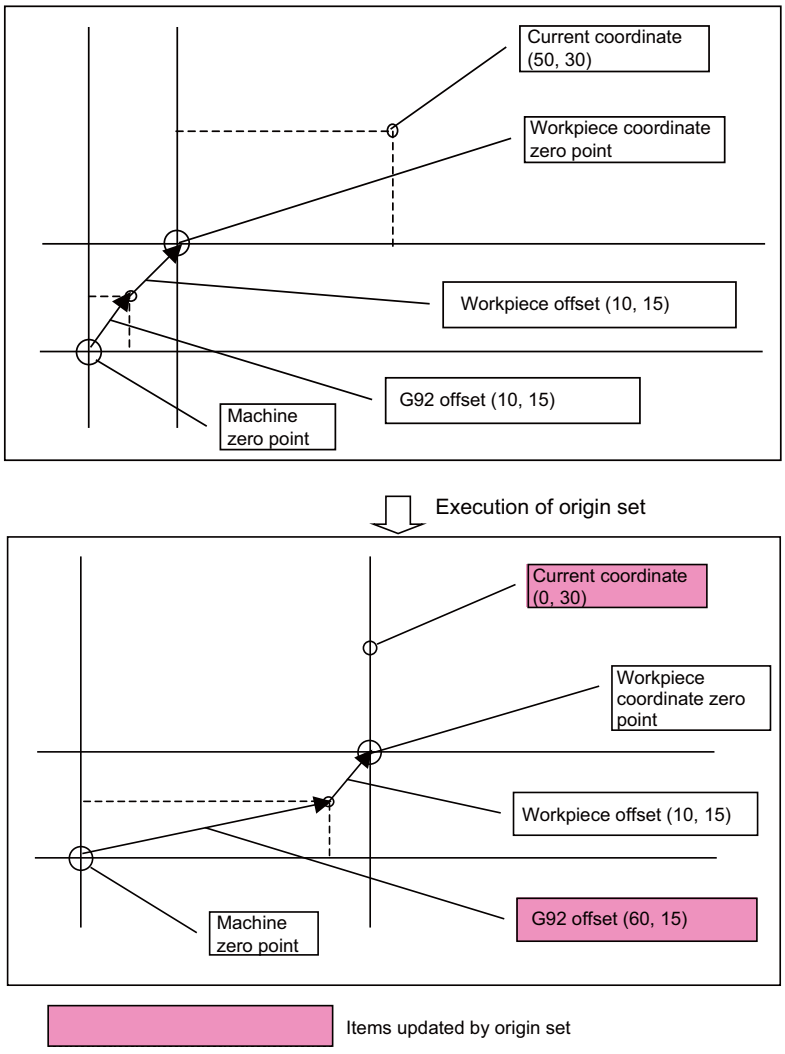
G19 X100.	With these program commands, X100. is the axis which does not exist on the G19 (Yp, Zp) plane, Yp-Zp plane is selected by G19 and the X axis moves by 100. mm separately from the plane selection.
G17 X100 . R50. ;	With these program commands, the Xp-Yp plane is selected by G17 and the arc command is controlled on the X-Y plane by this command.

10.1.10 Origin Set/Origin Cancel

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

Origin set

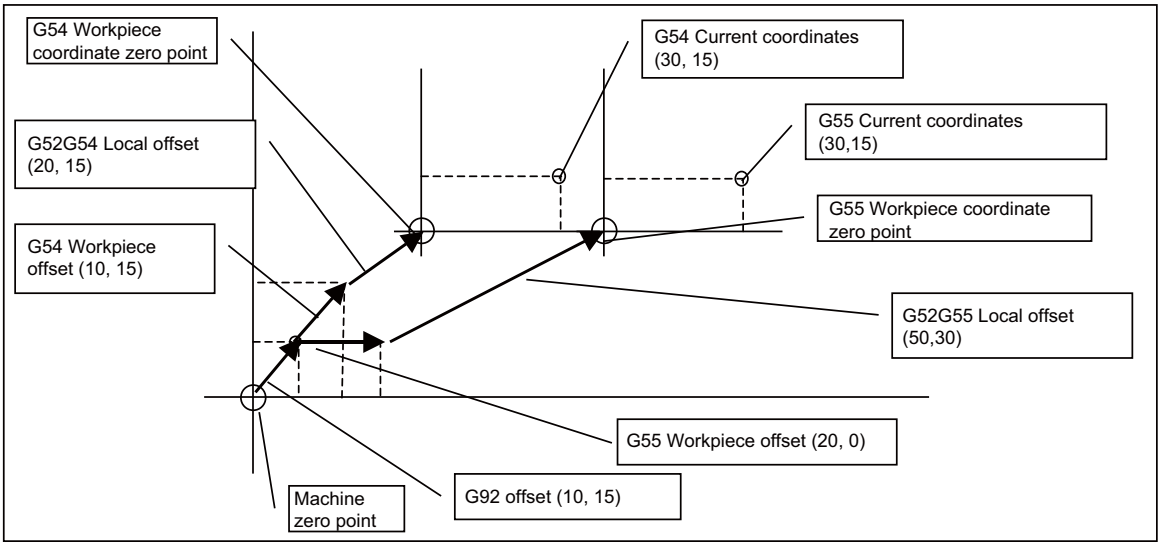
Origin set is a function that shifts the coordinate system so that the current position is the zero point on the workpiece coordinate system containing the workpiece coordinate system's offset value.
The relative position counter and workpiece coordinate counter are set to "0" with this operation. In other words, this is the same as the coordinate system setting command "G92 X0;". (For target axis: X)



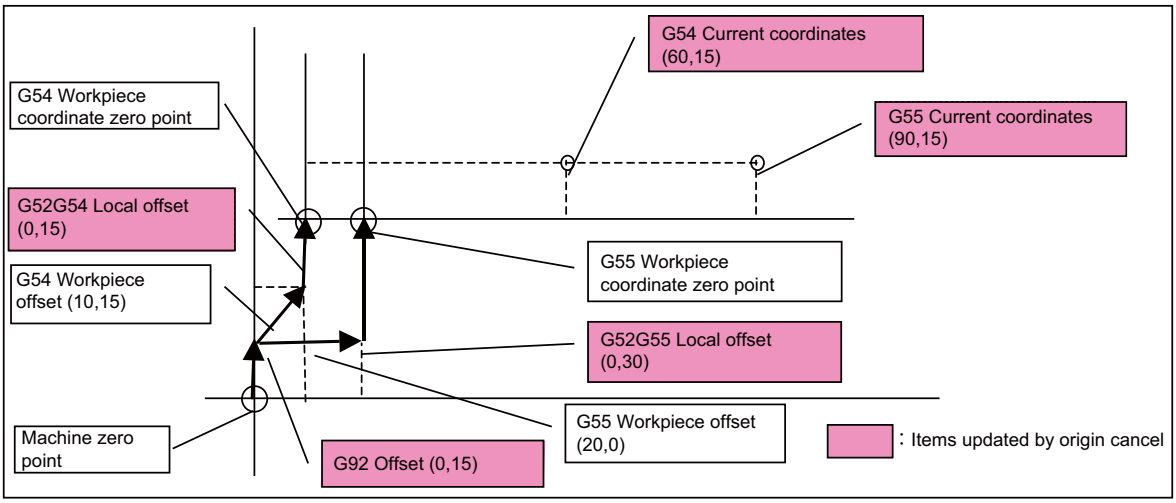
Origin cancel

Origin cancel is a function that manually cancels all deviated amounts, and shifts to the designated zero point with the workpiece offset.

The relative position counter and machine position counter are set to "0" with this operation. In other words, this is the same "G92 G53 X0 ;". (For target axis: X)



Execution of origin cancel



10.1.11 Counter Set

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

The relative position counter can be set to an arbitrary value by operating the setting and display unit screens.

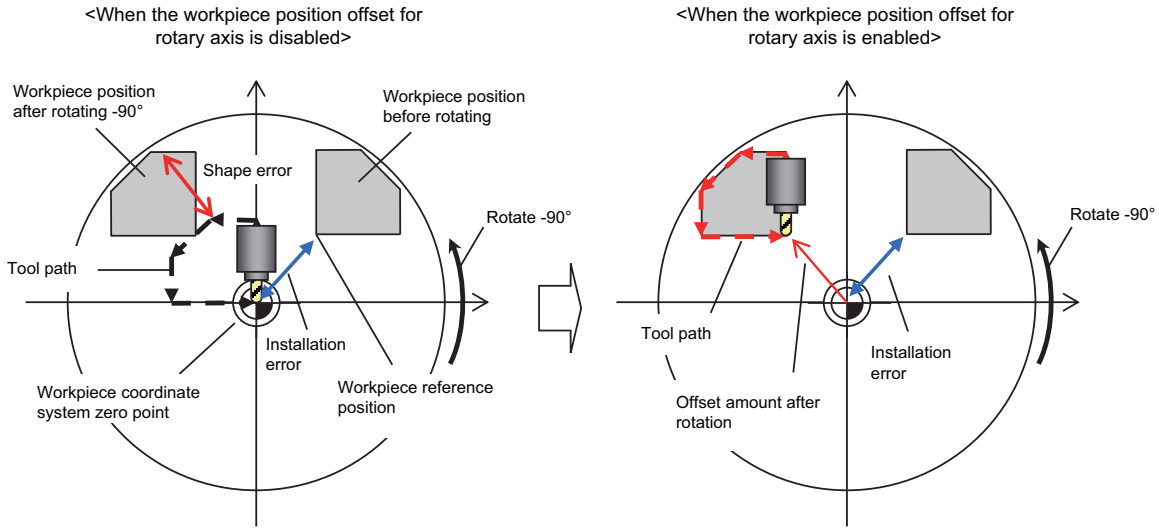
Select the axis and then input a value. Finally, press the INPUT key.

Only the [Relative Position] display field will change to the set value. The other coordinate positions displayed will not change. Up to 9 digits can be input in the integer section. The number of digits after the decimal point depends on the parameter setting.

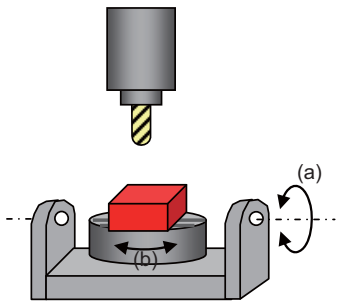
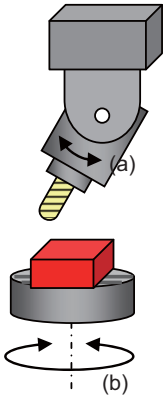
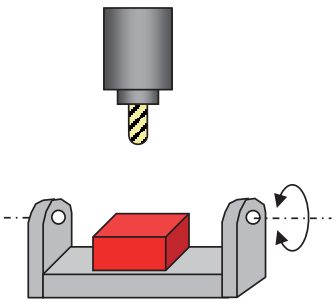
10.1.12 Workpiece Position Offset for Rotary Axis

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	—
L	—	—	—	—	—	—	—

For a machine equipped with a table rotary axis, this function corrects installation errors between workpiece coordinate zero point and workpiece reference position.
When a workpiece set on a rotary table is machined, if there is a difference between the workpiece coordinate zero point and the workpiece reference position, an error occurs in a machined shape. Setting the installation error between the workpiece coordinate zero point and the workpiece reference position (workpiece position offset amount for rotary axis) corrects in accordance with the table rotation angle for each block. This makes it possible to reduce the shape error at the time of machining, which enables machining with high accuracy.
This function is available only in the 1st part system. When this function is used in other than the 1st part system, an program error occurs.
The error is compensated for the axis set to be valid by the relevant parameter.



Workpiece position offset for rotary axis is valid in a machine configuration of the right-hand orthogonal coordinate system defined in ISO standard.
Workpiece position offset for rotary axis is applicable to the following three types of machine configuration:

Type	Table tilt type	Combined type	4-axis machine
Description	2 rotary axes on table side	A rotary axis each on tool head side and table side	3 orthogonal axes and 1 rotary axis
Example of machines			
Primary rotary axis (a)	The 1st rotary axis on the table side	Rotary axis on the tool side	Not applicable
Secondary rotary axis (b)	The 2nd rotary axis on the table side	Rotary axis on the table side	Not applicable

Note

(1) This function cannot be applied to a machine whose tool direction from the tool tip to the base is not parallel to the Z axis (Z axis positive direction) when machine position of all the rotary axes is 0 degree.
(2) This function cannot be applied to a machine whose rotation center axis of rotary axis is not parallel to any of the orthogonal coordinate axes.

■ Command format

Workpiece position offset for rotary axis

G54.2 P__ ;	
P	: Set No. of workpiece position offset for rotary axis (n = 0 to 8)

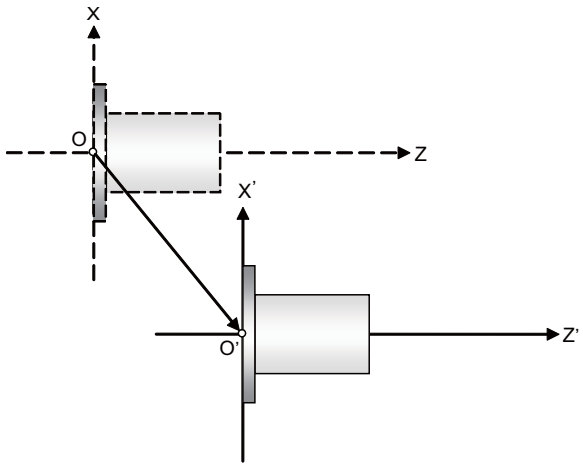
- Make sure to command G54.2 alone in a block. When it is commanded in the same block with other G codes, travel commands and miscellaneous commands, an program error occurs.
- When address P is not commanded, an alarm occurs. Also, commanding values other than "0" to "8" to the address P causes an alarm.
- If P is given in a block that contains no G54.2 command, the address P is not regarded as a command for specifying the workpiece position offset No. for rotary axis.
- G54.2 is a modal command of the G group 23.
- When "G54.2 Pm" is given during "G54.2 Pn", offset is applied with the offset set No. "m".
- Set the compensation amount of the group specified with address P to system variables. Refer to the "Programing Manual" for system variables for setting compensation amount.

10.1.13 Workpiece Coordinate System Shift

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	O	O	O	O	O	O	O

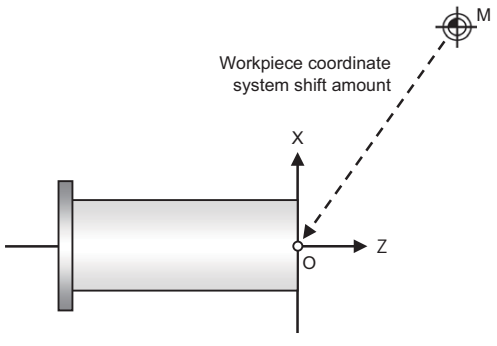
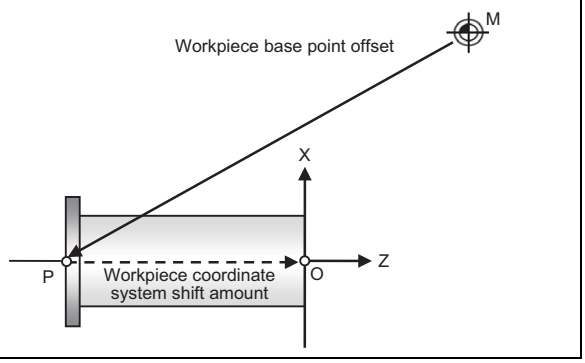
This function allows you to shift the workpiece coordinate system.
A workpiece coordinate system which was considered at machining programming may be misaligned with a coordinate system which is actually set or which is set by automatic coordinate system setting. The workpiece coordinate system at the program creation can be shifted to the measured workpiece coordinate system so that the machining can be performed without modification of the machining program. This function also eliminates the need to change the machining program even when the workpiece shape is changed.

[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

[Set the workpiece shape change amount as the workpiece coordinate system shift value.]
The base point of the shift amount of the workpiece coordinate system can be set to an arbitrary position. (This depends on the MTB specifications. (parameter setting))

Without workpiece base point offset	With workpiece base point offset
	

M: Basic machine coordinate zero point
P: Base point of shift amount of workpiece coordinate system (arbitrary position)
O: Coordinate system zero point at programming

The methods for the setting are as follows:
- Setting ON screen
- Setting with machining program (G code command)
- Setting by automatic measurement
- Setting and acquiring with system variables

10.2 Return

10.2.1 Manual Reference Position Return

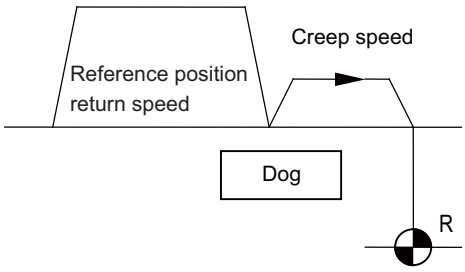
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

This function enables the tool to be returned manually to the position (reference position) which is characteristic to the machine.

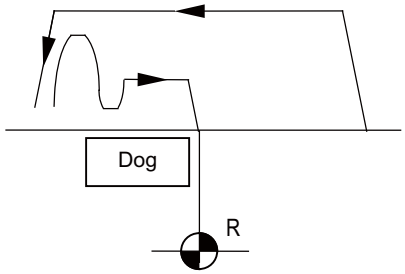
■ Return pattern to reference position

[Dog type]

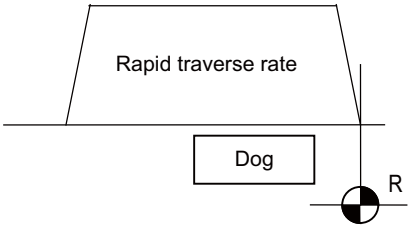
When starting in same direction as final advance direction



When starting in opposite direction as final advance direction



[High-speed type]



■ Differences according to detection method

	First return after power ON	Second return and following
Incremental position detection method	Dog-type	High-speed type
Absolute position detection method	High-speed type	High-speed type

10.2.2 Automatic 1st Reference Position Return

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

The machine can be returned to the first reference position by assigning the G28 command during automatic operation. If the interim point is commanded, the machine is moved up to that point by rapid traverse so that it is positioned and then returned separately for each axis to the first reference position.
Alternatively, by assigning the G29 command, the machine can be first positioned separately for each axis at the G28 or G30 interim point, and then positioned at the assigned position.

G code	Function
G28	Automatic 1st reference position return
G29	Start position return (The tool first returns to the interim position from the 1st reference position, and then is positioned at the position assigned in the program.)

■ Command format

G28 command format is as follows.

[M system]

```
G28 X__ Y__ Z__ α__ ;
```

G28	Automatic 1st reference position return
X, Y, Z, α	Coordinate value of interim point ("α" is the additional axis.)

[L system]

```
G28 X/U__ Z/W__ α__ ;
```

G28	Automatic 1st reference position return
X/U, Z/W, α	Coordinate value of interim point ("α" is the additional axis.)

G29 command format is as follows.

[M system]

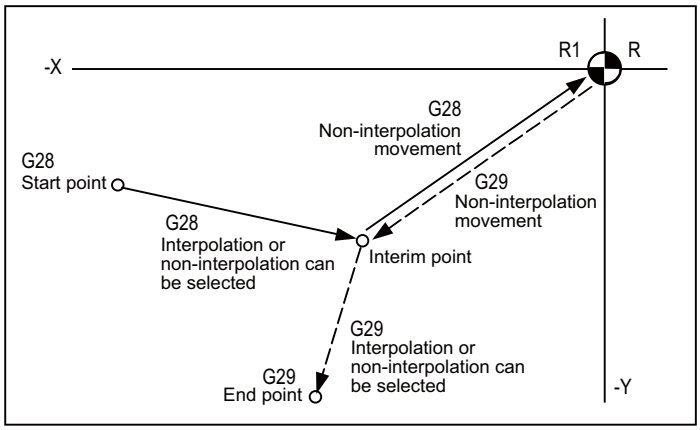
```
G29 X__ Y__ Z__ α__ ;
```

G29	Start position return
X, Y, Z, α	Coordinate value of end point ("α" is the additional axis.)

[L system]

```
G29 X/U__ Z/W__ α__ ;
```

G29	Start position return
X/U, Z/W, α	Coordinate value of end point ("α" is the additional axis.)



R1: 1st reference position

Interpolation or non-interpolation can be selected using a parameter for the movement up to the G28 interim point or for the movement from the G29 interim point to the command point. Non-interpolation applies for movement from the G28 interim point to the reference position and movement up to the G29 interim point.

If the position detector is for the incremental detection system, the first reference position return for the first time after the NC power has been turned ON will be the dog-type. However, whether the second and subsequent returns are to be the dog type or the high-speed type can be selected by designating a parameter.
The high-speed type is always used when the position detector is for the absolute position detection system.

Note

(1) The automatic 1st reference position return pattern is the same as for manual reference position return.
(2) The number of axes for which the 1st reference position return can be performed simultaneously depends on the number of simultaneous contouring control axes.
(3) If, at the time of the first reference position return, the tool radius compensation or nose radius compensation has not been canceled, it will be temporarily canceled during the movement to the interim point. The compensation is restored at the next movement after the return.
(4) If the tool length compensation has not been canceled, the interim point is the position where the compensation is applied.
In the machining center system, the tool length compensation is canceled and compensation amount is cleared after reference position return is completed. (*1)
In the lathe system, the tool length compensation is canceled temporarily during reference position return.
(5) When a single block operation is selected, it can be chosen by parameter setting whether or not to enable interim point stop.

(*1) The tool length compensation can be canceled temporarily by the setting of the parameter. In this case, the tool length compensation is restored by the next movement command.

10.2.3 2nd, 3rd, and 4th Reference Position Return

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

As with automatic 1st reference position return, commanding G30 P2 (P3, P4) during automatic operation enables the tool to be returned to the set points (2nd, 3rd or 4th reference positions) characteristic to the machine. Each reference position can be set by parameters.
The tool is first positioned by rapid traverse to the assigned interim point and then is returned to the reference position for each axis independently.

G code	Function
G30 P2	2nd reference position return
G30 P3	3rd reference position return
G30 P4	4th reference position return

■ Command format

G30 command format is as follows.

[M system]

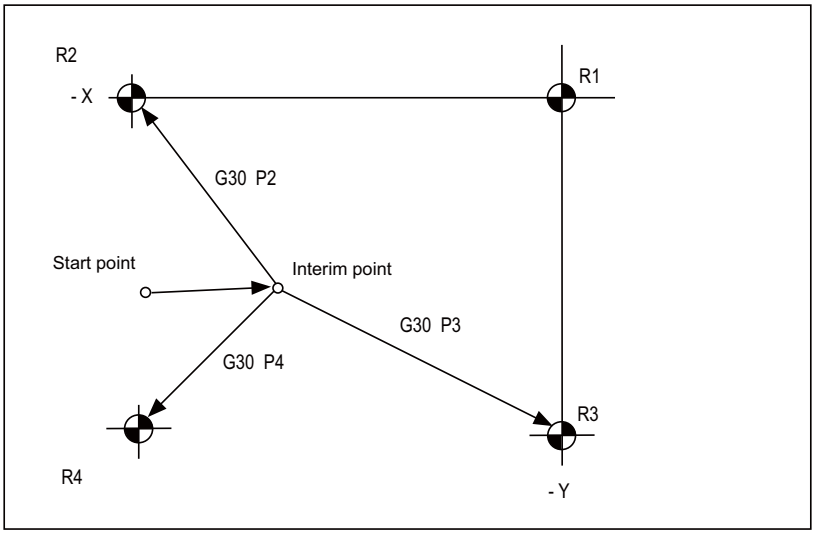
G30 P2 (P3, P4) X__ Y__ Z__ α__ ;

G30	Return command
P2 (P3, P4)	Return position No.
X, Y, Z, α	Coordinate value of interim point ("α" is the additional axis.)

[L system]

G30 P2 (P3, P4) X/U__ Z/W__ α__;

G30	Return command
P2 (P3, P4)	Return position No.
X/U, Z/W, α	Coordinate value of interim point ("α" is the additional axis.)



R1: 1st reference position
R2: 2nd reference position
R3: 3rd reference position
R4: 4th reference position

Note

(1) The second reference position return is performed if the P address is omitted.
(2) The number of axes for which reference position return can be performed simultaneously depends on the number of simultaneous contouring control axes.
(3) If, at the time of the reference position return, the tool radius compensation or nose R compensation has not been canceled, it will be temporarily canceled during the movement up to the interim point. The compensation is restored at the next movement after the return.
(4) If the tool length compensation has not been canceled, the interim point is the position where the compensation is applied.
In the machining center system, the tool length compensation is canceled and compensation amount is cleared after reference position return is completed. (*1)
In the lathe system, the tool length compensation is canceled temporarily during reference position return.
(5) Whether interpolation or non-interpolation is to apply to the movement up to the interim point can be selected using a parameter. Non-interpolation applies for movement from the interim point to each of the reference positions.
(6) When a single block operation is selected, it can be chosen by parameter setting whether or not to enable interim point stop.

(*1) The tool length compensation can be canceled temporarily by the setting of the parameter. In this case, the tool length compensation is restored by the next movement command.

10.2.4 Reference Position Check

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

By commanding G27, a machining program, which has been prepared so that the tool starts off from the reference position and returns to the reference position, can be checked to see whether the tool will return properly to the reference position.

■ Command format

G27 Xx1 Yy1 Zz1 Pp1 ;

G27	: Check command
Xx1,Yy1,Zz1	: Return control axes
Pp1	: Check No.
	P1:1st reference position check
	P2: 2
	P3: 3
	P4: 4

The tool is first positioned by rapid traverse to the assigned position and then, if this is the reference position, the reference position arrival signal is output.
When the address P is omitted, the first reference position check will be applied.

Note

(1) The number of axes for which reference position check can be performed simultaneously depends on the number of simultaneous contouring control axes.
(2) An alarm results unless the tool is positioned at the reference position upon completion of the command.
(3) Whether interpolation or non-interpolation is to apply to the movement can be selected using a parameter.

10.2.5 Absolute Position Detection

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

The absolute position detection function holds the position data of the actual machine position and the machine coordinates in the controller even after the power is turned OFF. When the power is turned ON again, a position command can be executed in automatic operation immediately without executing reference position return. (High-speed return is always used for the reference position return command.)
There are two types of absolute position detection method: dog type and dog-less type, which use different reference position establishment methods.
This is specified by the parameter.

Method			Details	Establishment of zero point	Adjustment of zero point position
Dog-less type	Machine end stopper method		The zero point is established by pressing the machine against a set point on the machine.	The zero point is established when a torque limit is applied on the servo and the torque limit is reached by pressing against the machine stopper.	Input a value equivalent to the shift amount on the absolute position setting screen.
	Basic position alignment method	Method I	The zero point position is set from the screen.	The zero point is established by inputting it through the absolute position setting screen.	Input a value equivalent to the shift amount on the absolute position setting screen.
		Method II			
Dog-type			Same method as incremental detection dog-type.	The zero point is established with dog-type reference position return completion.	Set a value in the parameter.

Note

(1) This function is valid for the NC axis and the PLC axis. This function cannot be used for the spindle and the auxiliary axis.

10.2.6 Tool Exchange Position Return

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

By specifying the tool change position in a parameter and also assigning a tool change position return command in a machining program, the tool can be changed at the most appropriate position.
The axes for which returning to the tool change position is performed and the order in which the axes begin to return can be changed by commands.

G30.n ;
n = 1 to 6: Specify the axes that return to the tool change position and the order in which they return. (For L system, n = 1 to 5)

■ Command and return order

[M system]

Command	Return order
G30. 1	Z axis -> X axis/Y axis (-> additional axis)
G30. 2	Z axis -> X axis -> Y axis (-> additional axis)
G30. 3	Z axis -> Y axis -> X axis (-> additional axis)
G30. 4	X axis -> Y axis/Z axis (-> additional axis)
G30. 5	Y axis -> X axis/Z axis (-> additional axis)
G30. 6	X axis/Y axis/Z axis (-> additional axis)

[L system]

Command	Return order
G30. 1	X axis only (-> additional axis)
G30. 2	Z axis only (-> additional axis)
G30. 3	X axis -> Z axis (-> additional axis)
G30. 4	Z axis -> X axis (-> additional axis)
G30. 5	X axis/Z axis (-> additional axis)

Note

(1) An arrow (->) indicates the order of axes that begin to return. A period (/) indicates that the axes begin to return simultaneously.
Example : "Z axis -> X axis" indicate that the Z axis returns to the tool change position, then the X axis does.
(2) G30.6 is only for the M system.

The tool change position return ON/OFF for the additional axis can be set with parameter for the additional axis.
For the order to return to the tool change position, the axes return after the standard axis completes the return to the tool change position (refer to above table).
The additional axis cannot return to the tool change position alone.

11

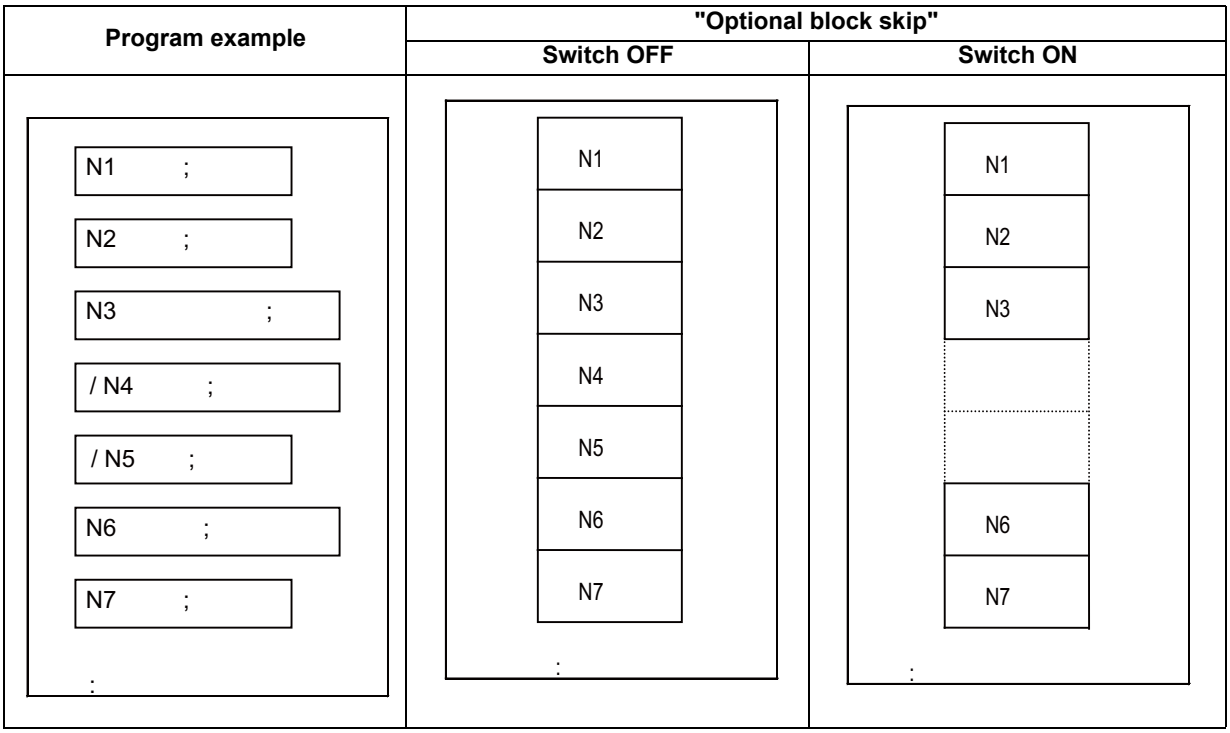
Operation Support Functions

11.1 Program Control

11.1.1 Optional Block Skip

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

It is possible to command to selectively ignore a part, from a "/" (slash) code to the end of the block, of a machining program. When the optional block skip input signal from the external source is turned ON for automatic operation, the block with the "/" code is skipped.
If the "Optional block skip" signal is turned OFF, the block with the "/" code will be executed without being skipped.



Skipping from a "/" in the middle of a program to the end of the program can also be enabled by parameter setting.

11.1.2 Optional Block Skip Addition

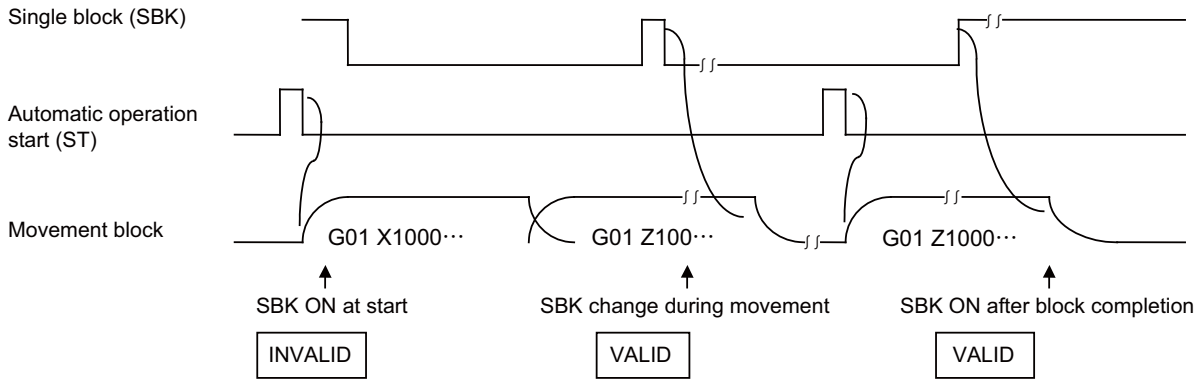
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

When "/n" (n: 1 to 9, slant code) is programmed at the head of a block, and the optional block skip n input signal from the external source is turned ON for automatic operation, the block with the "/n" code is skipped.
If the "Optional block skip n" signal is turned OFF, the block with the "/n" code will be executed without being skipped.

11.1.3 Single Block

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

The commands for automatic operation can be executed one block at a time (block stop) by turning ON the "Single block input" signal. When the single block input signal is turned ON temporarily during continuous operation, the machine will stop after that block has been executed.
Even when operation is switched to another automatic operation mode (for example, memory operation mode to MDI operation mode) during continuous operation, the machine will stop after that block has been executed.



As with the multi-part system specification, the following function can be selected instead.

- Multi-part system single block (L system)
  This function is for executing single block operation while maintaining the synchronization between the part systems when two or more part systems are operated. When one part system has been stopped by single block stop, the other part systems are stopped by feed hold.

11.2 Program Test

11.2.1 Dry Run

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

F code feed commands for automatic operation can be switched to the manual feed rate data of the machine operation board by turning ON the "Dry run input" signal.

Command	Dry run switch ON	
	Rapid traverse selection switch OFF	Rapid traverse selection switch ON
G00,G27,G28,G29,G30,G60	Manual feedrate	Rapid traverse rate
G01,G02,G03	Manual feedrate	Cutting clamp speed

11.2.2 Machine Lock

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

When the machine lock input signal is set to ON, the NC operations can be executed without assigning commands to the NC axes.

The feed rate in the machine lock status is the command speed.
The M, S, T and B commands are executed as usual and operation is completed by returning the FIN signal.

- Reference position return (manual, G28, G29, G30) is controlled as far as the interim point in the machine lock status but the block is completed when the interim point is reached.
- Machine lock is effective in the signal status applying when the axis has stopped.
- Block stop will be applied if the machine lock signal is turned ON to OFF or OFF to ON during automatic operation.

11.2.3 Miscellaneous Function Lock

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

The M, S, T and B (2nd miscellaneous function) output signals are not output to the machine or PLC when the "Miscellaneous function lock" signal of external input is turned ON. This function can be used when checking only the movement commands in a program check.
The start signals of the M command are output for the M00, M01, M02 and M30 commands, and so a completion signal must be returned.

- Fixed cycle spindle functions containing an S code and any M, S, T or B function assigned by a manual numerical command or in automatic operation will not be executed. The code data and strobe (MF, SF, TF, BF) outputs are stopped.
- If this signal is set ON after the code data has already been output, the output is executed as it would normally be executed until the end (until FIN1 or FIN2 is received and the strobe is turned OFF).
- Even when this signal is ON, the M00, M01, M02 and M30 commands among the miscellaneous functions are executed, and the decode signal, code data and strobe signals are also output as they would be normally.
- Any miscellaneous functions which are executed only inside the controller and not output (M96, M97, M98, M99) are executed as they would be normally even if this signal is ON.

11.2.4 Graphic Check

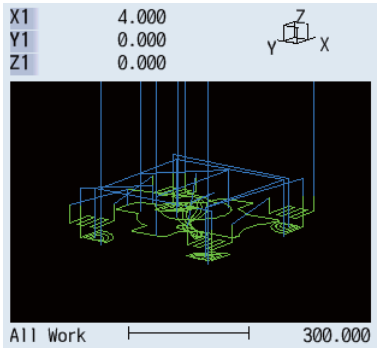
11.2.4.1 Graphic Check

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

The machining program movement path can be confirmed with the graphic drawing without executing automatic operation. Therefore, the machining program can be checked while drawing the figure accurately at a high speed.
There are two description method for the graphic check function: 2D check and 3D check.
The drawing's viewpoint can be moved, enlarged and reduced while drawing. The three axes displayed are set with the parameters.

Note

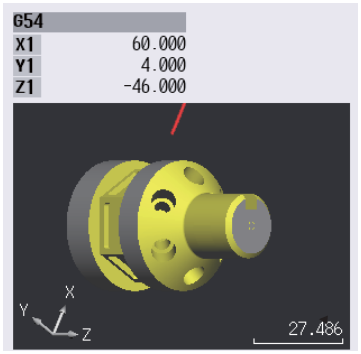
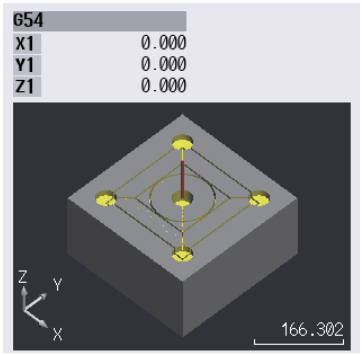
(1) Automatic operation is disabled during graphic check for M80V.



11.2.4.2 3D Solid Program Check

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

The machining program movement path can be confirmed with the graphic drawing without executing automatic operation. Therefore, the machining program can be checked while drawing the figure accurately at a high speed.
There are two types of check methods for the 3D check: Check method I and Check method II. The Check method I simplifies the machining result of machining program and simulates at a high speed. The check method II simulates the machining result of machining program at a higher accuracy than the Check method I.
The workpiece shape's viewpoint can be moved, enlarged and reduced while drawing. The three axes displayed are set with the parameters.
The workpiece shape and tool shape used are set on this screen.

[Drawing image (L system)]	[Drawing image (M system)]
	

11.2.4.3 Graphic Check Rotary Axis Drawing

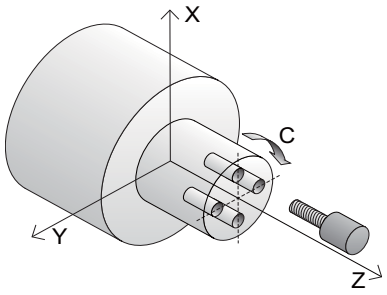
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	Δ	Δ	Δ	Δ	○	○	○

When the rotary axis (C axis) is designated with the parameter in program check (2D), the path is drawn depending on the rotary axis. The rotary axis can be set for each part system.
For details on the graphic check, refer to " 11.2.4.1 Graphic Check".

Note

(1) When the control axis address other than C axis is designated with the parameter, the path corresponding to the rotary axis is not drawn.
(2) The workpiece coordinate system must be set so that the center of the rotary axis becomes the coordinate system zero point.

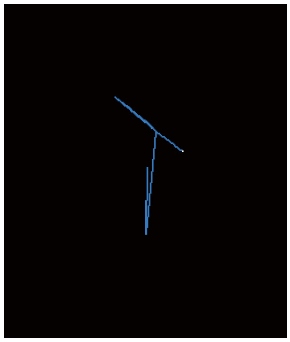
Image of actual tool and workpiece



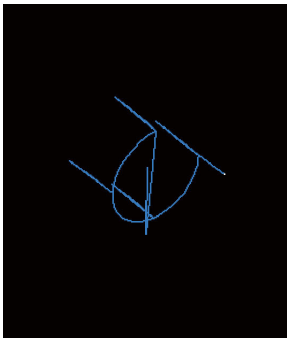
Example of machining program

```
G83Z-20.R-5.Q5000F300 ;
G0C90. ;
G83Z-20.R-5.Q5000F300 ;
G0C180. ;
G83Z-20.R-5.Q5000F300 ;
G0C270. ;
G83Z-20.R-5.Q5000F300 ;
G80 ;
```

Drawing image when the rotary axis drawing is invalid



Drawing image when the rotary axis drawing is valid



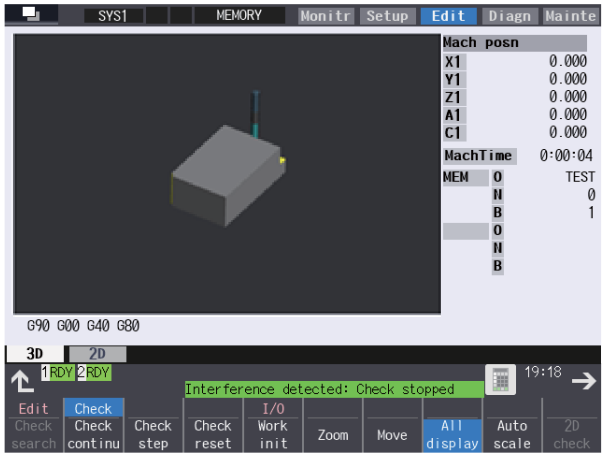
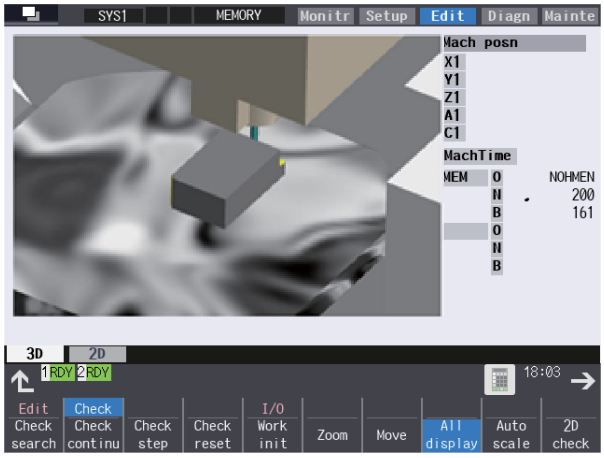
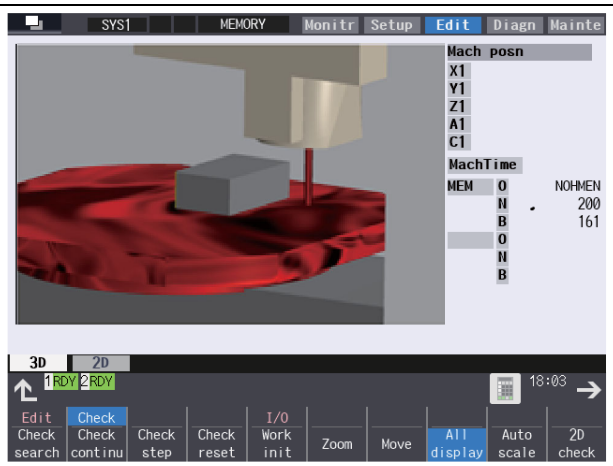
11.2.4.4 3D Machining Simulation

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	—	—	—	—	—
L	—	—	—	—	—	—	—

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.
Interference check is performed between the following items:

- Tool and other tool
- Tool and workpiece
- Tool and machine parts
- Workpiece and machine parts
- Machine parts and other machine parts

When the machine simulation is not supported, interference between the non-cutting part of the tool and the workpiece is checked.
Workpiece simulation and machine simulation can be switched.
There are two types of check methods: Check method I and Check method II. Check method I simplifies the machining result and simulates it at high speed. Check method II simulates the machining result with higher accuracy than Check method I.

[Workpiece simulation]	[Machine simulation]
	
[Interference check]	
	

11.2.5 Graphic Trace

11.2.5.1 Graphic Trace

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

This function draws the machine tool's machine position. The movement path following the actual automatic operation or manual operation, or the tool tip movement path can be drawn. In addition, the machine operation during machining can be monitored.
For the display mode, 1-plane, 2-plane and 3-dimensional display are provided. In the 3-dimensional display mode, cubic shapes can be rotated and tracing of the figure seen from the desired direction can be assigned.

One of the following functions is required to validate this function.
- Tool length compensation along the tool axis
- Tool handle feed & interruption
- Tool center point control
- 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

11.2.5.2 Graphic Trace Rotary Axis Drawing

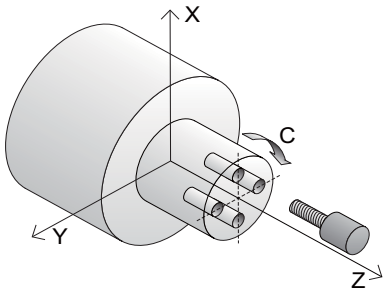
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	Δ	Δ	Δ	Δ	○	○	○

When the rotary axis (C axis) is designated with the parameter in program trace, the path is drawn depending on the rotary axis. The rotary axis can be set for each part system.
For details on the graphic trace, refer to " 11.2.5.1 Graphic Trace".

Note

(1) When the control axis address other than C axis is designated with the parameter, the path corresponding to the rotary axis is not drawn.
(2) The workpiece coordinate system must be set so that the center of the rotary axis becomes the coordinate system zero point.

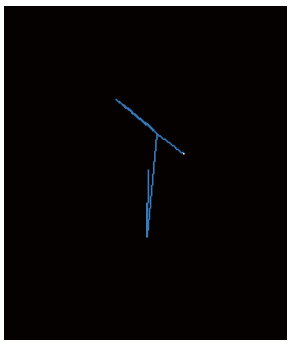
Image of actual tool and workpiece



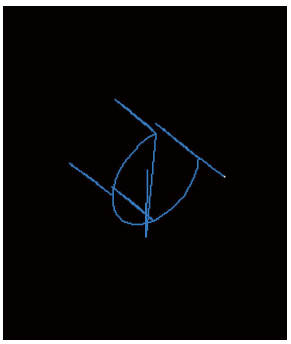
Example of machining program

```
G83Z-20.R-5.Q5000F300 ;
G0C90. ;
G83Z-20.R-5.Q5000F300 ;
G0C180. ;
G83Z-20.R-5.Q5000F300 ;
G0C270. ;
G83Z-20.R-5.Q5000F300 ;
G80 ;
```

Drawing image when the rotary axis drawing is invalid



Drawing image when the rotary axis drawing is valid



11.2.6 Machining Time Calculation

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

This function analyzes the machining program without moving the axis and calculates the approximate time required for machining.
Machining time is calculated by accumulating the following times:

- Rapid traverse time
- Cutting feed time
- Dwell time
- MSTB time

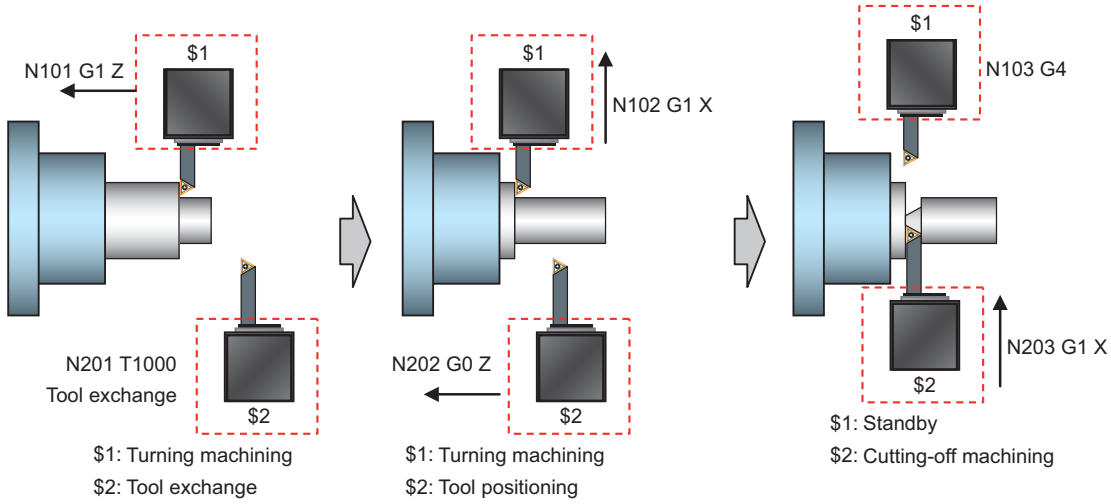
The machining time is displayed as "hour: minute: second".

11.2.7 Manual Arbitrary Reverse Run (Program Check Operation)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

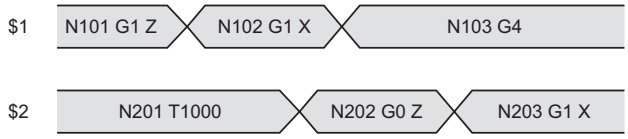
The manual arbitrary reverse run can be performed by controlling the feedrate being in the automatic operation in the memory or MDI mode in proportion to the manual feedrate by jog or the rotation speed by manual handle.
This function allows a program to run the executed blocks backward (reverse run) after the block stop in the automatic operation. The buffer correction of the program can be performed after reverse run to a desired point and the corrected program can be run.

■ Example of machining operation

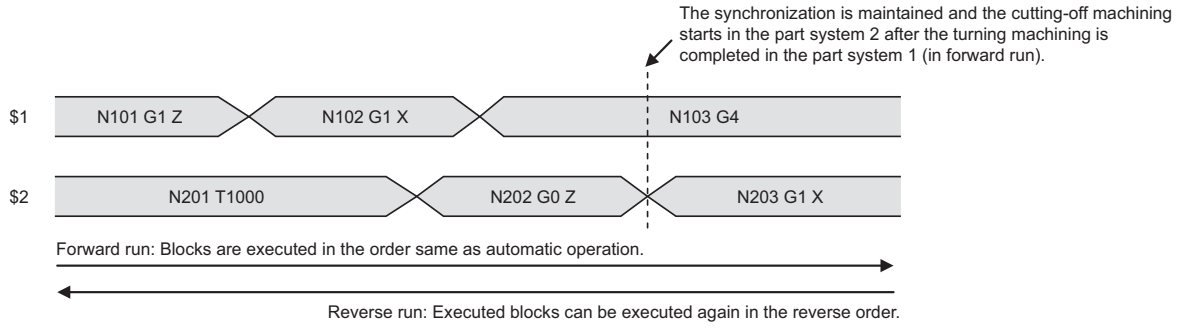


$1: Part system 1
$2: Part system 2

[Timing chart in override 100%]



[Timing chart in manual arbitrary reverse run]

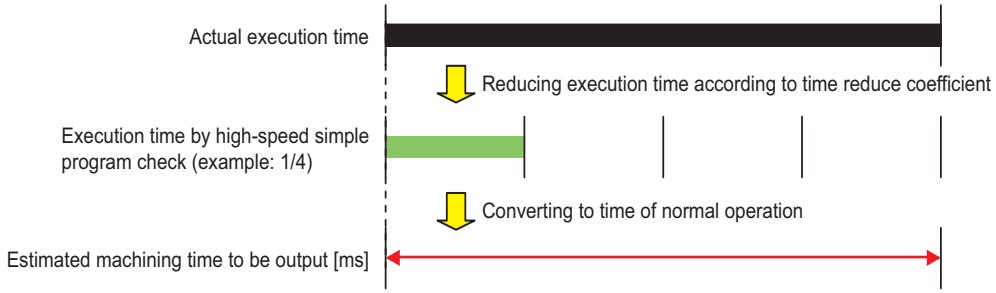


The total operation time is increased in the timing diagram for the manual arbitrary reverse run, however the synchronization between part systems can be maintained and the cutting-off machining starts after the turning machining is completed in forward run. The synchronization between part systems is maintained in both forward and reverse run.

11.2.8 High-Speed Simple Program Check

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

This function checks whether a program error occurs by operating the machining program without the axes movements.
The estimated machining time can be checked in time shorter than the actual execution time of the machining program.
This function is valid with the "High-speed simple program check" signal ON .
The machining program is operated in the reduced time according to the coefficient which is set to the time reduce coefficient.
The estimated machining time of the machining program is output to time measurement output.
The miscellaneous command code which outputs strobes can be allocated arbitrarily with the parameter during the high-speed simple program check.



11.3 Program Search/Start/Stop

11.3.1 Program Search

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

The program No. of the program to be operated automatically can be designated and called. Upon completion of search, the head of the program searched is displayed.
Machining programs are stored in the memory inside the NC system.

11.3.2 Sequence Number Search

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

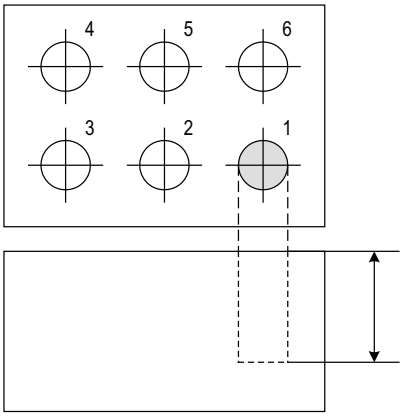
Blocks can be indexed by setting the program No., sequence No. and block No. of the program to be operated automatically. The searched program is displayed upon completion of the search.
Machining programs are stored in the memory inside the NC system.

11.3.3 Collation Stop

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

This function enables the single block stop status to be established at any block without having to set the SINGLE BLOCK switch to ON.
It can be used to readily check the machining shape up to the designated block and resume machining.

```
    G91 ;
    :
    G00 Z-150. ;
    N100 G81 X-100. Z-100. R-50. F100 ;
->  N101 X-100. ;
    N102 X-100. ;
    N103 Y100. ;
    N104 X100. ;
    N105 X-100. ;
    :
```



Collation and stop is executed when drilling of one hole is completed (N100 block is completed) as shown on the left.
The hole depth is measured to confirm that the dimensions are correct. If the dimensions are not correct, the tool compensation amount is changed.
If machining is started from the head of the program, accurate hole drilling can be carried out.

11.3.4 Program Restart

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

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.
The machine will not move during the search.
When the search is completed, the distance from the current position of the machine (workpiece coordinates) to the restart position (start point of the searched block) is displayed as restart remaining distance on the screen.
After the program restart, positioning to the restart position is carried out at the cycle start, and automatic operation is performed for the searched and subsequent blocks.
Positioning to the restart position can also be manually performed.
After the program restart, an arbitrary M, S, T or B command can be executed by the manual numerical value command before the cycle start.

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 search methods by the program restart.

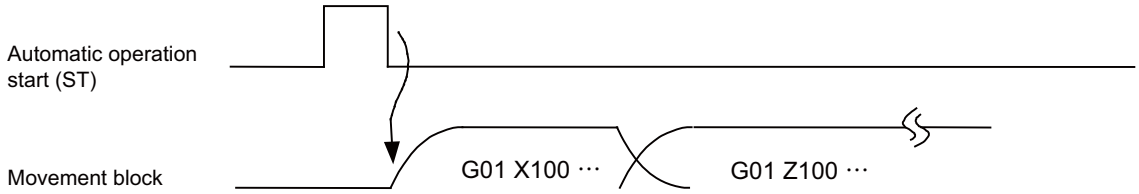
Restart method	Details	
Restart search type 1	After machining is reset due to a tool breakage, etc., machining is restarted from the designated sequence number and/or block number. Only the program which had been executed just before can be restarted. Even after the power is turned ON again, the program can be restarted if the program has been executed before the power supply is turned OFF.	
Restart search type 2	If a machining program different from the machining program to be restarted was run before starting restart search, specify the sequence No. and block No. before restarting the machining.	
	Automatic top search OFF	A top search must be executed from the screen. Then, command a sequence No. and block No., and restart a program.
	Automatic top search ON	A top search may not be executed from the screen. A top search is automatically executed, so it is not necessary to execute a top search from the screen. Command a program No., sequence No., and block No., and restart a machining. An arbitrary program can be restarted by designating the program No. When a program No. is omitted, the program currently searched is restarted.

The validity of the automatic top search can be changed with control parameter.

11.3.5 Automatic Operation Start

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

With the input of the "Automatic operation start" signal (change from ON to OFF), the automatic operation of the program that has been operation searched is started by the controller (or the halted program is restarted).



Automatic operation startup is performed on a part system by part system basis.

11.3.6 NC Reset

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

This function enables the controller to be reset.

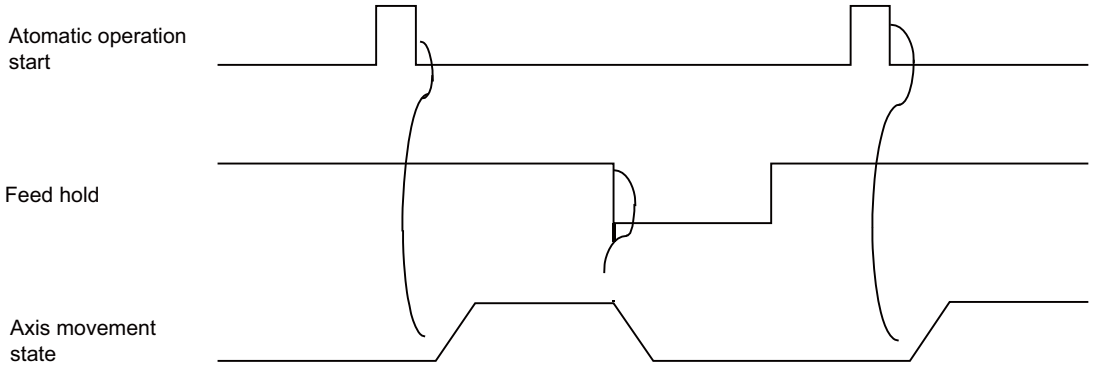
		Reset 1	Reset 2	Reset & Rewind
(1)	**G command modals**	Retained	Initialized	Initialized
(2)	**Tool compensation data**	Retained	Cancel (no operations)	Cancel
(3)	**Memory indexing**	Executed	Not executed	Executed
(4)	**Errors/alarms**	Reset	Reset	Reset
(5)	**M, S and T code outputs**	Retained	Retained	Retained
(6)	**M code independent output**	OFF	OFF	OFF
(7)	**Control axis moving**	Deceleration stop	Deceleration stop	Deceleration stop
(8)	**Output signals**	"In reset" signal	"In reset" signal	"In reset" signal "In rewind" signal

11.3.7 Feed Hold

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

When the feed hold signal is set ON during automatic operation, the machine feed is immediately decelerated and stopped. The machine is started again by the "Automatic operation start (cycle start)" signal.

- When the feed hold mode is entered during cycle start, the machine feed is stopped immediately, but the M, S, T and B commands in the same block are still executed as programmed.
- When the mode is switched during automatic operation to manual operation (such as jog feed, handle feed or incremental feed), the feed hold stop mode is entered.
- An interrupt operation based on manual operation (such as jog feed, handle feed or incremental feed) can be executed during feed hold.



11.3.8 Search & Start

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

If the "Search & start" signal is input in a status where the memory mode is selected, the designated machining program is searched and executed from its head.
If the "Search & start" signal has been input during automatic operation in the memory mode, search & start is executed after resetting.

11.3.10 Auto-restart

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

A machining program is restarted automatically at the completion of the machining program execution.

11.4 Interrupt Operation

11.4.1 Manual Interruption

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

Manual interrupt is a function that enables manual operations to be performed during automatic operation.
The methods to select the operation mode are as follows:

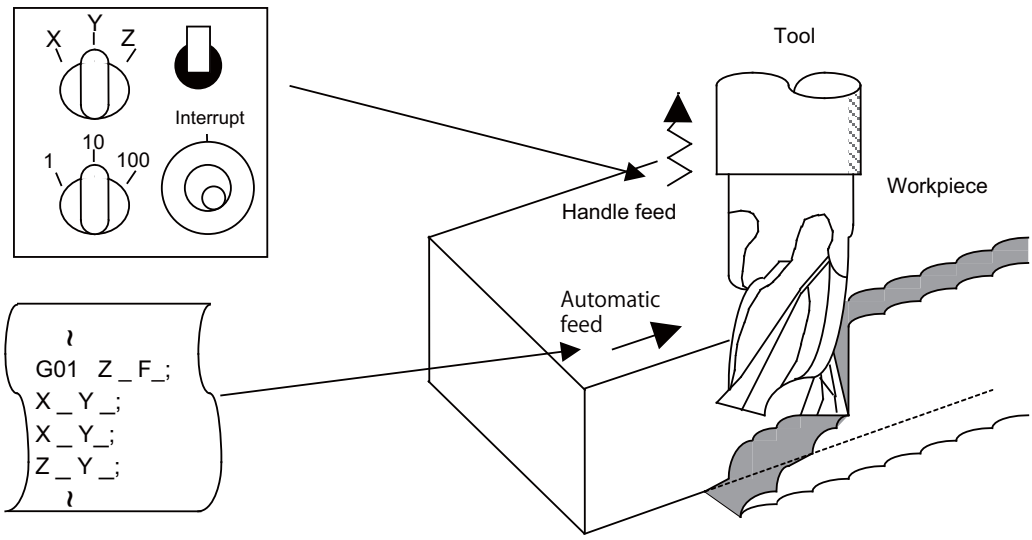
- Initiates the interrupt by switching from the automatic operation mode to manual operation mode
- Initiates the interrupt by selecting the manual operation mode at the same time as the automatic operation mode

(Refer to " 11.4.9 Simultaneous Operation of Manual and Automatic Modes".)
Whether the manual interrupt amount is to be retained and automatic operation is to be continued is determined by setting manual absolute mode ON or OFF (refer to " 11.4.3 Manual Absolute Switch").

11.4.2 Automatic Operation Handle Interruption

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

The handle command can interrupt and be superimposed onto a command without suspending automatic operation and the machine can be moved by rotating the manual pulse generator during automatic operation.
If the spindle load is greatly exceeded when cutting a workpiece as per the machining program due to a high rough cutting amount in face machining, for instance, automatic handle interrupt makes it possible to raise the Z surface and reduce the load easily without suspending feed in the automatic operation mode.
Automatic handle interrupt is conducted by setting the "automatic handle interrupt" valid switch which is provided separately from the "manual operation mode". The axis selection, pulse scale factor and operation are conducted as for manual handle feed.
Whether, after an interrupt, to return to the path of the machining program by automatic operation or remain offset by the amount equivalent to the interrupt amount is determined using a parameter.

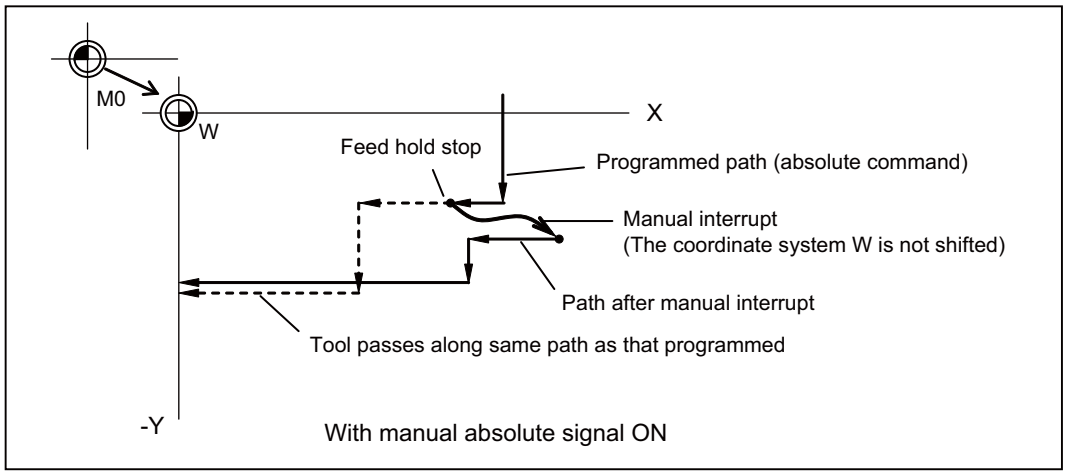


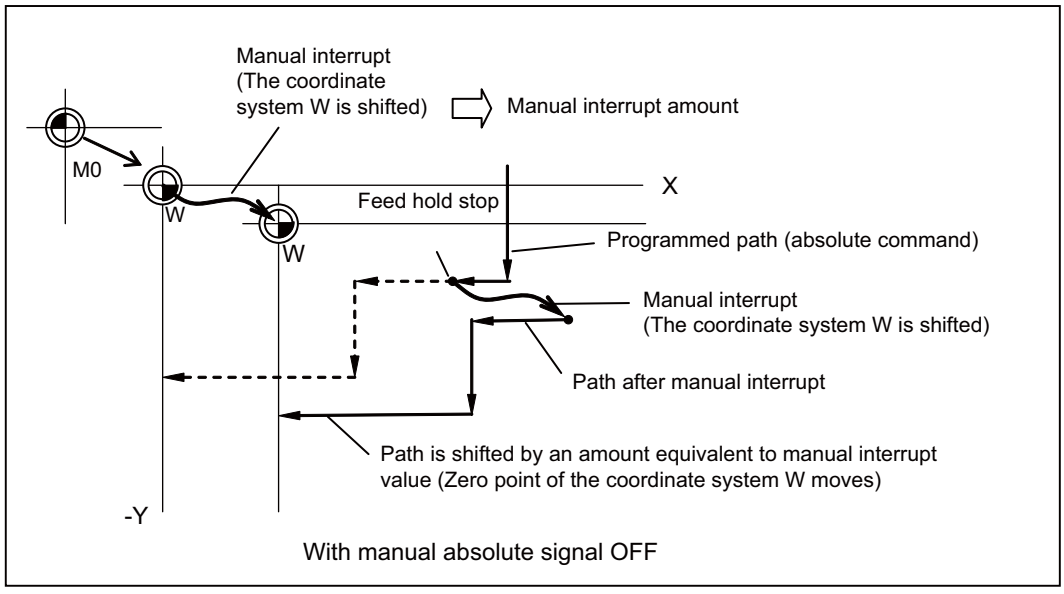
Feed path with automatic feed and handle feed superimposed

11.4.3 Manual Absolute Switch

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

The program absolute positions are updated by an amount equivalent to the distance by which the tool is moved by hand when the manual absolute switch signal is turned ON.
In other words, the coordinate system based on the original program will not shift even if the tool (machine) is moved by hand. Thus, if automatic operation is started in this case, the tool will return to the path before manual movement.



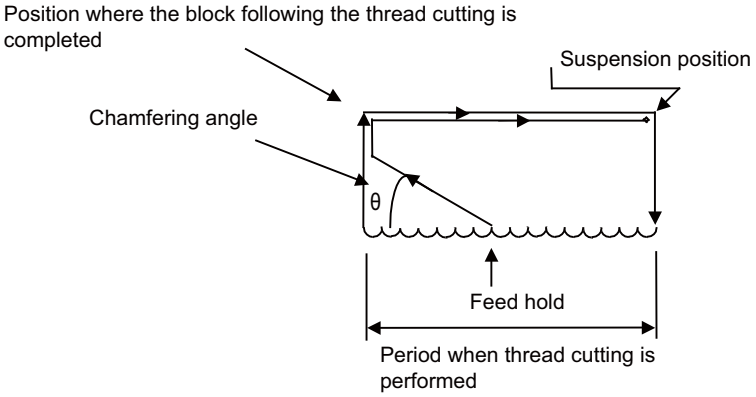


The manual absolute ON state will be entered when the power is turned ON.

11.4.4 Thread Cutting Cycle Retract

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	—
L	O	O	O	O	O	O	O

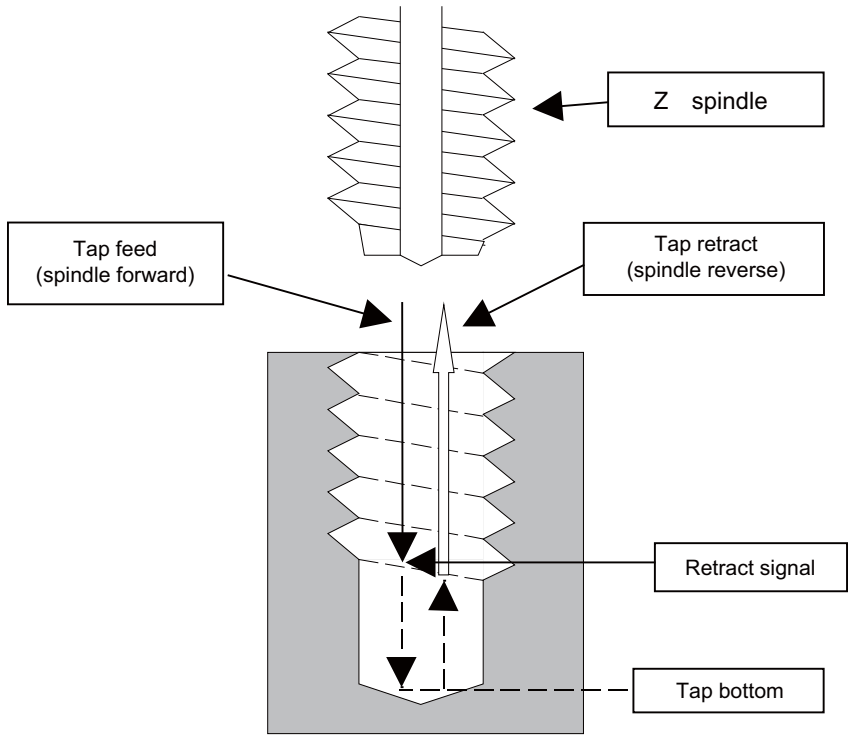
This function suspends the thread cutting cycle if a feed hold signal has been input during thread cutting in a thread cutting cycle.
If a feed hold signal is input during chamfering or thread cutting without chamfering, operation stops at the position where the block following the thread cutting is completed.



11.4.5 Tapping Retract

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

If tapping is interrupted by a reset or emergency stop signal that is input during tapping and the tap is left engaged inside the workpiece, the tap tool engaged inside the workpiece can be rotated in the reverse direction so that it will be disengaged by inputting the "Tap retract" signal.



This function can be used when the machining was interrupted upon reset, emergency stop or power OFF during tapping cycle. (Note that "power OFF" is applied only when the absolute position detection system is valid.)
A return is made to the initial point by tap retract.

11.4.6 Manual Numerical Value Command

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

Commands of spindle function (S), miscellaneous function (M), tool function (T) and 2nd miscellaneous function (B) can be executed through the Monitor screen or Setup screen.
This enables operations such as spindle speed changing, starting, stopping, calling and selecting assigned tools and replacing of the spindle tools to be done easily without having to prepare or revise the machining program.
The corresponding macro program can be executed from the manual numerical value command.

Miscellaneous command macro call	Calls and executes the macro program registered as an M code macro (*1) from the M command.
T command macro call	Calls and executes the T code macro program registered exclusively for manual numerical value command from the T command.

(*1) The M code macros that can be used for manual numerical value command are limited to those with a call type of M98.

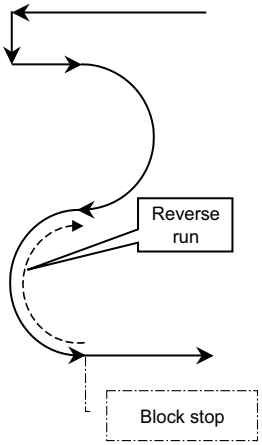
11.4.7 Arbitrary Reverse Run

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	—
L	—	—	—	—	—	—	—

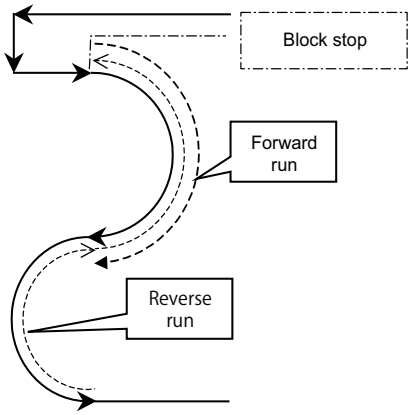
This function allows a program to run the executed blocks backward (hereinafter called "reverse run") after the block stop in the automatic operation.
Perform the reverse run as shown below.
(1) Execute the reverse run to the point to go back.
(2) Run the reversed blocks again following the program (hereinafter called "forward run").
(3) Continue executing the remaining blocks from the interrupted position.

Maximum 200 blocks can be executed in the reverse run.
Only the 1st to 3rd axes of each part system can be used for this function.

[Reverse run]



[Forward run after the reverse run]



11.4.8 MDI Interruption

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

This function enables MDI programs to be executed during automatic operation in the single block stop status. When the modal status is changed in the MDI program, the modal status in the automatic operation mode is also changed.

11.4.9 Simultaneous Operation of Manual and Automatic Modes

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

This function enables manual operations to be performed during automatic operation by selecting an automatic operation mode (tape, MDI or memory) and manual mode (handle, step, jog or manual reference position return) simultaneously. (Arbitrary feed based on the PLC is also possible.)

The feed rates for the axes subject to automatic commands and the feed rates for axes subject to manual command are set separately. The acceleration/deceleration modes (rapid traverse, cutting feed) are also set separately. Rapid traverse override, cutting feed override and 2nd cutting feed override are valid both for axes subject to automatic commands and manual commands. Override cancel is valid for axes subject to automatic commands.
Manual interlock is applied to axes subject to manual commands; automatic interlock is applied to axes subject to automatic commands.

11.4.10 Simultaneous Operation of JOG and Handle Modes

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

When executing the jog feed and handle feed, both these feeds are available without changing the mode each time by inputting the jog mode signal and simultaneous operation of jog and handle modes signal to the control unit. However, while axis is moving in one of the two modes, the feed in the other mode is not valid.

11.4.11 Reference Position Retract

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

When the retract signal is turned ON during the automatic and manual operation, this function can retract the tool immediately to a set reference position.
The reference position to be retracted to can be selected from the 1st reference position to 4th reference position with 2-bit input signal.
Set the retracting order of axes with parameter.

■ Other operations

- When the retract signal is turned ON, the control unit is reset, the operation is interrupted, and the machining program is indexed.
- When the "Rapid traverse input" signal is input, the rapid traverse rate is applied. When the "Rapid traverse input" signal is not input, the manual feed rate is applied.
- If the retract signal is input during execution of a tapping cycle, the operation will be the tapping retract, and the normal reference position retract will be executed from the end point of tapping retract operation.
- Even if the retract signal is input during the thread cutting cycle, it will be invalid. However, if the retract signal is input in a block other than the thread cutting block, the retracting operation will be executed.
- If the retract signal is turned OFF midway during retracting, the operation will decelerate and stop. However, since the machining program is indexed, the block can not be resumed.
- The retract signal is invalid if the coordinate system is not established. An operation error will occur when the retract signal is input in such case.

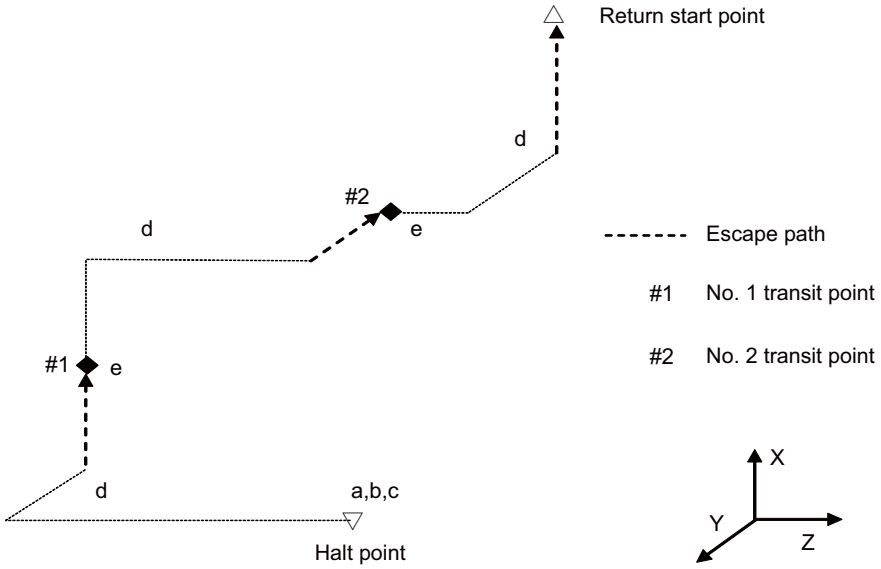
11.4.12 Tool Retract and Return

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

Even if the machining program's operation is halted and the tool is escaped to change the tool or check the workpiece, etc., the tool can be returned to the halted point (machining halted point) and machining resumed.

The two main functions are as follow.

- After the machining program is halted, the halt point is designated, and the tool is escaped manually.
  The transit points for returning are designated when escaping.
- The tool returns automatically.
  When returning, the tool passes through the transit points designated when escaping and returns to the halt point. Then, machining restarts.



(a) Machining program halt
(b) Halt point designation
(c) Operation mode changeover
(d) Tool retract
(e) Transit point designation

11.4.13 Skip Retract

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	—	—	—	—	—	—	—

This function is used to return to the opposite advance direction when the skip signal is input during G31 command.

11.4.14 PLC Interruption

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

The interrupt program set with the R register is executed with the signals from the PLC during single block stop in program operation or during the manual mode.

12

Program Support Functions

12.1 Machining Method Support Functions

12.1.1 Program

12.1.1.1 Subprogram Control [Layers]

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○ 10	○ 10	○ 10	○ 10	○ 10	○ 10	○ 10
L	○ 10	○ 10	○ 10	○ 10	○ 10	○ 10	○ 10

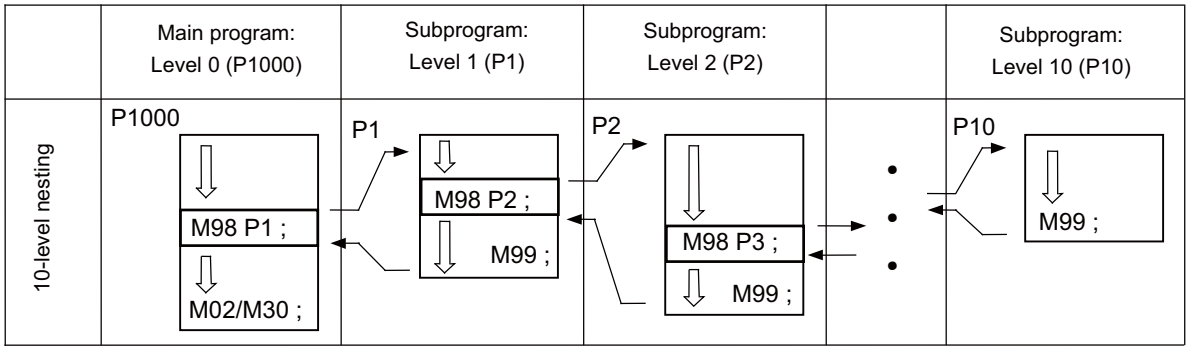
When the same pattern is repeated during machining, the machining pattern can be registered as one subprogram and the subprogram can be called from the main program as required, thereby realizing the same machining easily. Efficient use of program can be made. The subprogram is called with the program No. and sequence No. or the file name and sequence No.

Calling the subprogram with M98 command

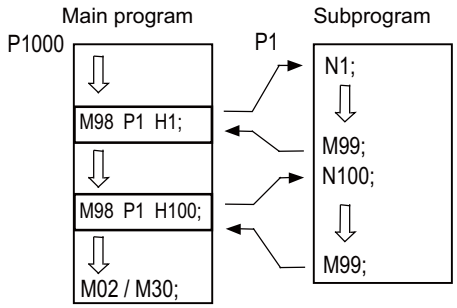
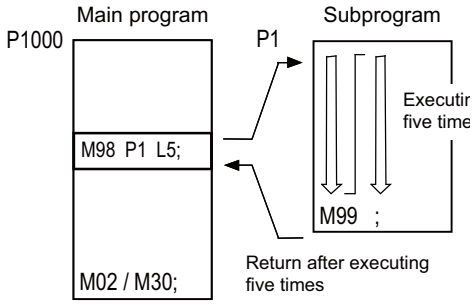
M98 P__ H__ L__ ,D__ ;
or, M98 <File name> H__ L__ ,D__ ;

M98	Subprogram call command
P	Program No. in subprogram to be called (Own program if omitted.) Note that "P" can be omitted only during memory mode operation (NC memory, NC memory 2, high-speed program server, SD card, hard disk, USB) and MDI operation. (Maximum 8-digit value) A four-digit or eight-digit program No. starting with O No. can be called by the parameter. However, the commanded subprogram is called when the number of digits of the command value is larger than the one set in the parameter.
<File name>	A file name can be designated instead of the program No. In this case, enclose the file name inside brackets < >. (The file name can have up to 32 characters, including the extension.) (Example) M98 <BUHIN-12.RAF>;
H	Sequence No. in subprogram to be called (Head block if omitted)
L	Number of subprogram repetitions (If omitted, this is handled as L1. When "L0" is set, the subprogram is not called.) (1 to 9999 times)
,D	Device No. where subprogram is stored. The setting of ",D" depends on the model.

Subprograms can be nested up to 10 levels deep. (The nesting depth depends on the model.)



A subprogram branch destination or repetition of a subprogram can be specified.

Specifying a subprogram branch destination	Specifying repetition of a subprogram
	

Returning to main program from the subprogram with M99 command

M99 P__ ;

P	Sequence No. of return destination (returned to block that follows the calling block)

Calling the subprogram with M198 command

A program registered on the SD card can be called as a subprogram. To call a program in the SD card as the subprogram, command as shown below in the main program.

```
M198 P__ L__ ;
or, M198 <File name>  L__ ;
```

M198	Call command
P	Program No. in SD card to be called as subprogram (Maximum 8 digits) A four-digit or eight-digit program No. starting with O No. can be called by the parameter. However, the commanded subprogram is called when the number of digits of the command value is larger than the one set in the parameter.
<File name>	A file name can be designated instead of the program No. In this case, enclose the file name inside brackets < >. (The file name can have up to 32 characters, including the extension.)
L	Number of subprogram repetitions (Maximum 4 digits) This can be omitted. (In this case, the subprogram will be called once.) When L0 is set, the subprogram is not called.

Note

(1) Sequence No. call (M198 H****) is not possible.
(2) The device that can be used for M198 subprogram call differs depending on the NC models.
M800VW/M80VW: Control unit-side SD card
M800VS/M80V: Front-side SD card
(3) Calling of the subprogram with the M198 command is limited to once in the subprogram nest.
The subprogram can be called only from the NC memory, NC memory 2 or MDI program.
(4) The section from the head of the program to the first LF (line feed code, 0x0A hexadecimal) is invalid, and is not run or displayed. Note that if the head starts from an O number (program No.), the program is valid from the head.
(5) A program registered in an SD card can be executed from only one part system. A program error will occur if an attempt is made to execute the programs in the SD card simultaneously by two or more part systems. If all the part system is reset when the error occurred, programs will be displayed as only "%" except for the first part system.
(6) For information on how to call a subprogram with "<File name>, O number", refer to "Calling the subprogram with M98 command".

Returning to main program from the subprogram with M99 command

```
M99 ;
```

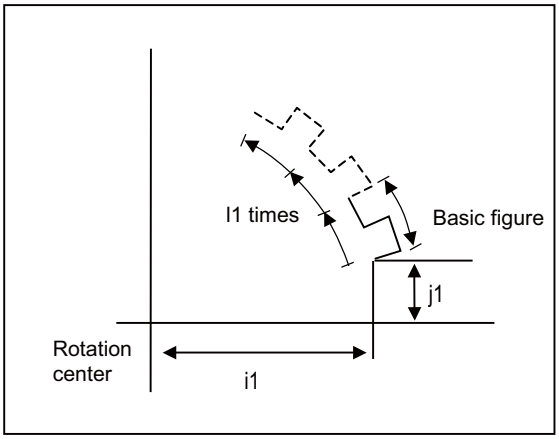
12.1.1.2 Figure Rotation

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	—	—	—
L	—	—	—	—	—	—	—

If the same pattern is used repeatedly on a concentric circle, one of the rotating machining patterns can be registered as a subprogram. When the subprogram is called from the main program, if the rotation center is designated, a path similar to the rotary phase can be easily created on the concentric circle. This simplifies creation of the program.
The program format is shown below.

M98 Ii1 Jj1 (Kk1) Pp1 Hh1 Ll1 ,Dd1;
or, M98 Ii1 Jj1 (Kk1) <File name> Hh1 Ll1 ,Dd1 ;

M98	Subprogram call command
Ii1 Jj1 (Kk1)	Rotation center coordinates
Pp1	Program No. in subprogram to be called (Own program if omitted.) Note that P can be omitted only during memory mode operation (NC memory, NC memory 2, high-speed program server, SD card, hard disk, USB) and MDI operation. (Maximum 8-digit value) A four-digit or eight-digit program No. starting with O No. can be called by the parameter.
<File name>	A file name can be designated instead of the program No. In this case, enclose the file name inside brackets < >. (The file name can have up to 32 characters, including the extension.) (Example) M98 <BUHIN-12. RAF> ;
Hh1	Sequence No. in subprogram to be called (Head block if omitted)
Ll1	Number of subprogram repetitions (If omitted, this is handled as L1. When "L0" is set, the subprogram is not called.) (1 to 9999 times)
,Dd1	Subprogram device No. (0 to 4) The subprogram in the memory can be used when ,D is omitted. The device No. is set with the machining parameters.



- The first subprogram called out with subprogram call is executed at a 0° rotation angle. The path is created as commanded.
- If the number of repetitions is set to two or more, the rotation angle is obtained from the called subprogram's start point, end point and rotation center coordinate. The path of the first subprogram is used as a basic figure and is rotated and arranged for the designated number of call repetitions, using the rotation center coordinates as a reference.
- During figure rotation, all blocks in the subprogram are treated as to be rotated.
- If the subprogram start point and end point are not on the same circle having the commanded figure rotation center coordinates as the center, the axis will interpolate using the subprogram's end point as the start point, and the end point in the first movement command block in the rotated subprogram as the end point.
- The figure is rotated on the workpiece coordinate system, and can be shifted with the G92, G52, G54 to G59 (workpiece coordinate system shift) command.
- Functions (reference position return, uni-direction positioning, etc.) on the machine coordinate system for the rotary plane axis cannot be used while the figure is rotated. However, the machine coordinate system functions can be used for axes other than the rotation plane.
- A program error will occur if figure rotation is commanded during figure rotation.
- Figure rotation and coordinate rotation by program cannot be commanded simultaneously.

12.1.1.3 Scaling

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	—	—	—	—	—	—	—

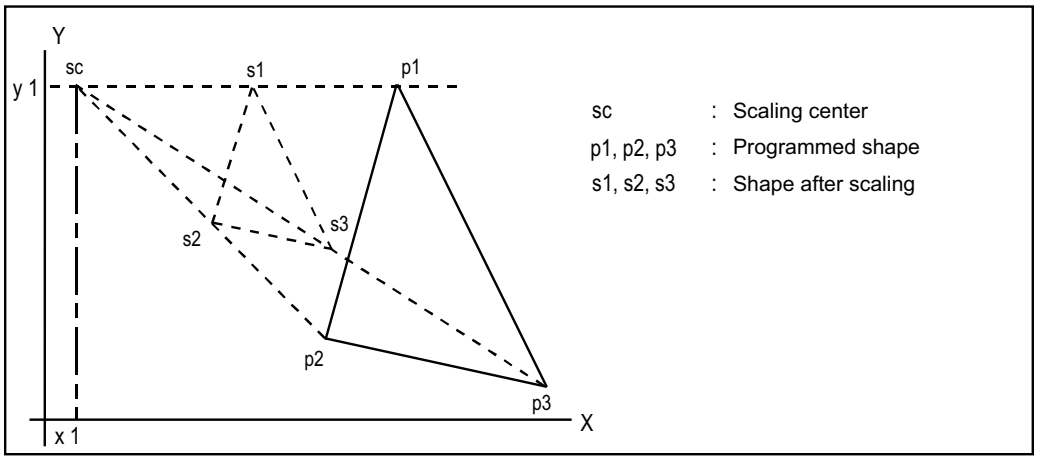
The shape commanded by the program can be extended or reduced to the desired size by applying a scale factor to the movement axis command position.

G code	Function
G50	Scaling cancel
G51	Scaling ON

■ Command format

G51 Xx1 Yy1 Zz1 Pp1 ;	
G51	: Call command
Xx1,Yy1,Zz1	: Scaling center coordinate position
Pp1	: Scale factor

The scale factor ranges from 0.000001 to 99.999999 times.



Note

(1) Scaling cannot be applied to compensation amount for tool radius compensation, tool position offset, or tool length compensation, etc. (The compensation and offset are calculated for the scaled shape.)

(2) Scaling applies only to the axes commanded with G51 block; it does not apply to axes which have not been commanded. When the scale factor is not assigned, the parameter setting applies instead.

12.1.1.4 Axis Name Switch

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	O	O	O	O	O	O	O

The axis name switch function switches the name of command axis and control axis.

G111 Axis name1 Axis name2 ;

G111	Axis name switch command
Axis name 1,2	Axes to be performed the name switch command When the axis name switch command is canceled, assign only "G111;" command.

When switching the name of X-axis and Y-axis by axis name switch command, the machine performs as follow.

<Before axis name switch>			**<After axis name switch>**	
Command axis name	Control axis		Command axis name	Control axis
X	X axis	⇨	X	Y axis
Y	Y axis		Y	X axis

(Example)
G111 X Y ;
G01 X100. ; (Y axis moves to 100.mm)
G01 Y100. ; (X axis moves to 100.mm)

12.1.2 Macro Program

12.1.2.1 User Macro [Layers]

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○ 4	○ 4	○ 4	○ 4	○ 4	○ 4	○ 4
L	○ 4	○ 4	○ 4	○ 4	○ 4	○ 4	○ 4

Macro commands 1 ; G65 to G67

In order to carry through one integrated function, a group of control and arithmetic instructions can be used and registered as a macro program. Furthermore, subprograms with a high degree of expandability can be configured by setting these macro programs as types which are capable of conducting control and arithmetic operations using variable commands.

G code	Function
G65	Macro call (Sample call)
G66	Macro modal call A
G66.1	Macro modal call B
G67	Macro modal call cancel

The program formats are given below

G65 P__ L__ Argument ;
or G65 <File name> L__ Argument ;

G65	: Call command
P	: Program No. A four-digit or eight-digit program No. starting with O No. can be called by the parameter.
<File name>	: File name When designating the file name, enclose the file name in angle brackets < >.
L	: Number of repeats
Argument	: Variable data designation

The macro program is called immediately by this command

G66 P__ L__ Argument ;
or G66 <File name> L__ Argument ;

G66	: Call command
P	: Program No. A four-digit or eight-digit program No. starting with O No. can be called by the parameter.
<File name>	: File name When designating the file name, enclose the file name in angle brackets < >.
L	: Number of repeats
Argument	: Variable data designation

The macro program is executed from the block with the axis command following this command.

G66.1 P__ L__ Argument ; **or G66.1 <File name> L__ Argument ;**	
G66.1	: Call command
P	: Program No. A four-digit or eight-digit program No. starting with O No. can be called by the parameter.
<File name>	: File name When designating the file name, enclose the file name in angle brackets < >.
L	: Number of repeats
Argument	: Variable data designation

The macro program is executed with the word data of each block as the argument.

The following macro command functions are available.

Arithmetic commands	#1=<Expression> ; Various arithmetic operations can be conducted between variables by the above. <Expression> is a combination of constants, variables, functions and operators.	
Assignment of priority of arithmetic operations	The portion in which the operator is to be given priority can be enclosed in [ ]. Up to five pairs of square parentheses [ ] including the function [ ] can be used. The normal priority of operation is functions and multiplication/division followed by addition/subtraction.	
Control commands	(1) If[<Conditional expression>]GOTO n ; (2) WHILE[<Conditional expression>] Do m ; ・・・ END m ;	The flow of the program can be controlled by these commands. "n" denotes the sequence numbers of the branching destination. "m" is an identification number, and 1 to 127 can be used. Note that only 27 nestings can be used.

Note

(1) The variable commands are provided under the optional specifications independently of the user macros. If they are to be used, specify the optional specifications separately.

Macro commands 2

Specific G commands and the miscellaneous commands (M, S, T, B) can be used for macro call.

■ Macro call using G codes

Simply by assigning a G code, it is possible to call user macro programs with the prescribed program number.

[Command format]

G P__ L__ Argument;**	
G**	: G code for performing macro call
P	: Available when the macro argument L/P valid function is enabled. It is not available when the macro argument L/P valid function is disabled.
L	: Available when the macro argument L/P valid function is enabled. It is not available when the macro argument L/P valid function is disabled.
Argument	: Variable data designation

The correspondence between the G** code which performs macro call and the program number for the macro to be called is set by a parameter.

G codes and G codes with decimal points can be used for this function. Consecutive G code macros can be set all at once by designating the first G code, the first No. of the program, consecutive number of G codes, and macro call type.
Maximum of 538 G codes can be used.

- In the range of G01 to G9999, 10 G codes can be set individually, and 255 G codes can be set all at once.
- In the range of G0.1 to G999.9, 10 G codes can be set individually, and 255 G codes can be set all at once.
- 8 codes are set by parameters (G200, G300, G400, G500, G600, G700, G800 and G900)

Note

(1) The G codes that are used in the system also can be made available for this use by parameter setting.
(2) G101 to G110 and G200 to G202 are user macro I codes. However, if a parameter is set for the G code call code, the G code call will have the priority, and these cannot be used as the user macro I.

■ Macro call using miscellaneous commands (M, S, T, B code macro call)

Simply by designating an M (or S, T, B) code, it is possible to call user macro programs with the prescribed program number. (Entered M codes and all S, T and B codes can be used.)

[Command format]

M (or S**;, T**;, B**;) P__ L__ Argument;**	
M** (S**,T**,B**)	: M (or S, T, B) code for performing macro call
P	: Available when the macro argument L/P valid function is enabled. It is not available when the macro argument L/P valid function is disabled.
L	: Available when the macro argument L/P valid function is enabled. It is not available when the macro argument L/P valid function is disabled.

The correspondence between the M** code which performs macro call and the program number for the macro to be called is set by a parameter. Among M00 through M9999, 10 M codes can be registered individually, and the number of M codes specified by the parameter can be registered by batch setting. The batch setting of consecutive macro programs as many as specified is possible by specifying the first M code, the first program number, the number of consecutive programs, and the type of macro call with the parameters.
Select codes to be entered other than the codes basically required by the machine and the following M codes.

- M0, M1, M2, M30, M96 to M99, M198, and M codes for G83 (parameter setting)

Note

(1) G commands in G code macro programs are not subject to macro calls but normal G commands. M commands in M code macro programs are not subject to macro calls but normal M commands. (The same applies to S, T and B codes.)
(2) The registration of the program number used for calling the G code macro or M (or S, T, B) code macro can be done independently for each part system.

Macro argument L/P valid

This function allows the addresses L (number of subprogram repetitions) and P (calling program No.) used as commands in user macro to be used as arguments. This function is enabled by a parameter.

Followings can be used by user macro.

- G65 Macro call (Sample call)
- G66 Macro modal call A
- G66.1 Macro modal call B
- Macro call using G codes
- Macro call using miscellaneous commands
- ASCII macro

Common variables protection

This function protects the common variables used for machine tool builder macro from being changed by user operation. The common variables in the range designated with the parameter cannot be changed by user operation, such as through a machining program, screen operation or file input.
The range of common variables to be protected can be designated individually for each part system.

12.1.2.2 Machine Tool Builder Macro

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

This function enables dedicated macro programs made by MTB to be registered in the CNC. By registering the macro as a program numbered in the display, editing, and input/output of the macro programs can be protected by a password, and the confidentiality of the macro programs can be maintained.

Machine tool builder macro programs are stored in a dedicated area which means that the user program registration area is not reduced in the process.
When registering a machine tool builder macro, secure its space by formatting.

There are two types of macro program memory area; SRAM area and ROM area. They can be switched with the parameter. ROM area has larger capacity than SRAM area, and more macro programs can be saved.
To keep registering/editing macro programs in ROM area after the NC is turned OFF and ON, it is necessary to write them in ROM.

[Macro program memory capacity]
SRAM area

Model	When the simple programming is in use (*1)		When the simple programming is not in use	
	Memory capacity (KB)	Maximum registerable number of programs (programs)	Memory capacity (KB)	Maximum registerable number of programs (programs)
M800VW/M800VS	173	572	256	600
M80VW/M80V	125	172	208	200

ROM area

Model	When the simple programming is in use (*1)		When the simple programming is not in use	
	Memory capacity (KB)	Maximum registerable number of programs (programs)	Memory capacity (KB)	Maximum registerable number of programs (programs)
M800VW/M800VS	429/941 (*2)	772/972 (*2)	512/1024 (*2)	800/1000 (*2)
M80VW/M80V	429	772	512	800

(*1) When the simple programming is in use means that the simple programing (NAVI MILL/NAVI Lathe) is being used and the macro is stored in the machine tool builder macro area.
(*2) One of them is optionally selected.

Note

(1) The memory capacity and the maximum registerable number of programs vary depending on the simple programing macro when the simple programming macro is registered in the machine tool builder macro area.

MTB unique numbers can be registered as the password.
There are the following two methods to manage the password.
Type 1: The password is set with numbers ("2" to "99999999"). (R register setting)
Type 2: The password is set with alphanumeric characters (one to eight letters). (Parameter setting)
For Type 2, the password is not displayed on the screen, and the confidentiality of the machine tool builder macro programs is improved compared to Type 1.

The macro program registered as a machine tool builder macro is called out as "G65 command" or "G code macro equivalent to G65". The macro program is described in the same manner as a conventional machining program.
Variables for the machine tool builder macros can be used within the machine tool builder macros. These variables can be used commonly within the machine tool builder macro programs.

■ Command format

User macro call (G65)

G65 P__ L__ Argument ;	
P	: Machine tool builder macro program No.(100010000 to 199999998)
L	: Number of repetitions (0 to 9999)
Argument	: Variable data designation

Note

(1) The machine tool builder macro program cannot be called with G66, G66.1 or M98. Note that a machine tool builder macro program can be called from a machine tool builder macro program using M98.

Macro call with G code (G***)

G* Argument;**	
***	: G code defined in macro definition program or G command macro parameter
Argument	: Variable data designation Use this argument when a local variable needs to be passed on to the macro program. Designate a real value after the address.

Miscellaneous command macro call (M or T code macro call)

M* Argument; (or T***)**	
***	: M (or T) code defined in macro definition program
Argument	: Variable data designation Use this argument when a local variable needs to be passed on to the macro program. Designate a real value after the address.

Note

(1) If M96, M97, M98, M99, M198 or the M code for G83 S Mode (for M system only) is commanded, machine tool builder macro will not be called. The said M code will be handled as a normal miscellaneous command

12.1.2.3 Macro Interruption

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

This function inputs from the machine to interrupt the program currently being executed, and instead calls and executes another program.
The program can be operated flexibly enough to meet varying conditions.
There are two types of interrupts, type 1 and type 2, as described below, and they are selected using a parameter.

- [Interrupt type 1] The block being executed is immediately interrupted, and the interrupt program is run immediately.
- [Interrupt type 2] After the block being executed is complete, the interrupt program is executed.

■ Command format

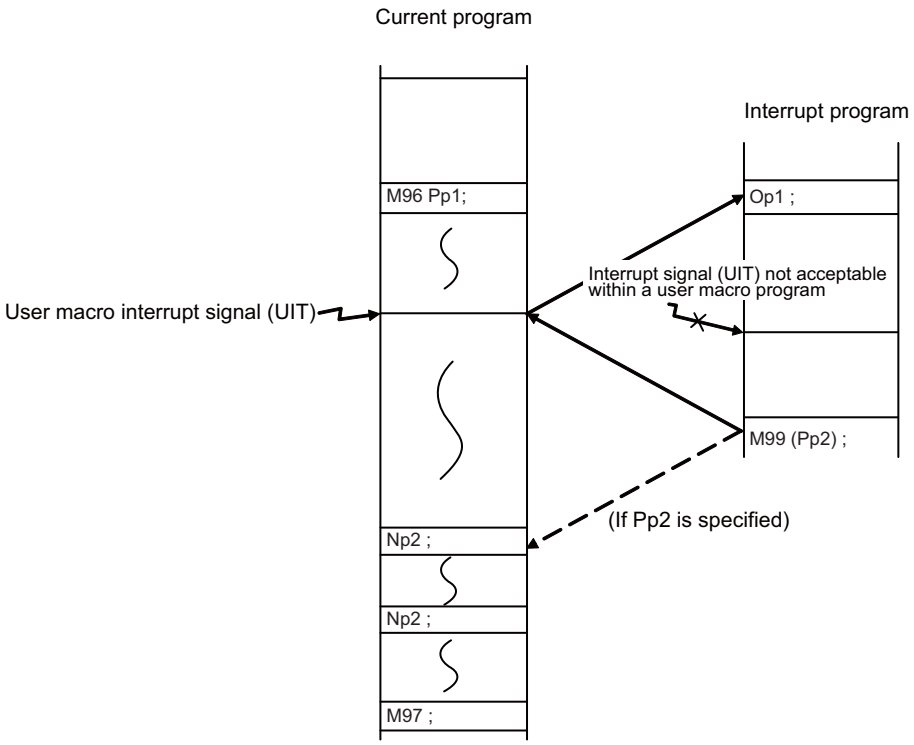
M96 P__ H__ ;	**User macro interrupt valid**

M96 <File name> H__ ;	**User macro interrupt valid**

M96	Interrupt valid command
P	Interrupt program number A four-digit or eight-digit program No. starting with O No. can be called by the parameter.
<File name>	File name When designating a file name, enclose the file name in angle brackets < >.
H	Interrupt sequence No.

M97 ;	**User macro interrupt invalid**

M97	User macro interrupt end command



12.1.2.4 Variable Commands

[M system]

Variable sets	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
600 sets	—	—	—	—	—	—	—
700 sets	○	○	○	○	○	○	○
8000 sets	△	△	△	△	○	○	—
(600 + 100 x number of part systems) sets	○	○	○	○	○	○	—
(7900 + 100 x number of part systems) sets	△	△	△	△	○	○	—

[L system]

Variable sets	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
600 sets	—	—	—	—	—	—	○
700 sets	○	○	○	○	○	○	—
8000 sets	△	△	△	△	○	○	—
(600 + 100 x number of part systems) sets	○	○	○	○	○	○	—
(7900 + 100 x number of part systems) sets	△	△	△	△	○	○	—

Programming can be given flexible and general-purpose capabilities by designating variables instead of directly assigning numbers for addresses in programs and by supplying the values of those variables as required when running the programs. Arithmetic operations (adding, subtracting, multiplying and dividing) can also be conducted for the variables.

Number of variable sets specifications

Type			Number		Function
Common variable			Common variables 1	Common variables 2	Can be used commonly for main, sub and each macro program. When using the common variables across multiple part systems, Some MTB specifications allow you to designate the number of common variables that are common to part systems.
	For 1-part system specifications	200 sets	500 to 599	100 to 199	
		600 sets	500 to 999 100100 to 800199 (*1)	100 to 199	
		700 sets	400 to 999 100100 to 800199 (*1)	100 to 199	
		8000 sets	400 to 999 100100 to 800199 (*1) 900000 to 907399	100 to 199	
	For multi-part system specifications	600 + 100 sets	400 to 999 100100 to 800199 (*1)	100 to 199 × number of part systems	
		7900 + 100 sets	400 to 999 100100 to 800199 (*1) 900000 to 907399	100 to 199 × number of part systems	
Local variables			1 to 33		Can be used as local in macro program.
R device access variables			50000 to 50749 51000 to 51749 52000 to 52749		Can be read/written from PLC and GOT.
System variables			1000 to		Application is fixed in system.
Fixed cycle variables			1 to 32		Local variables in fixed cycle program.

(*1) The part system common variable which can be used is shown in the table below.

Variable sets		Common variables 1
Variable sets specification	600 sets	#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)
	700 sets (600 + 100 sets)	
	8000 sets (7900 + 100 sets)	

Note

(1) All common variables are held even when the power is turned OFF.
(2) The common variables can be emptied by resetting or turning the power OFF when the parameters are set accordingly.
(3) The common variables are divided into the following two types.
Common variable 1: Variables that can be used commonly through the part systems.
Common variable 2: Variables that can be used commonly within that part system's program.
(4) The variable names can be set for some common variables.
(5) The common variables used for machine tool builder macro can be protected from being changed by user operation.

[Variable expressions]

		Example
#m	m = value consisting of 0 to 9	#100
# [f]	f = one of the followings in the formula	# [-#120]
	Numerical value m	123
	Variable	#543
	Formula Operator Formula	#110+#119
	- (minus) formula	-#120
	[Formula]	[#119]
	Function [formula]	SIN[#110]

[Variable definition]
Variable = expression

Note

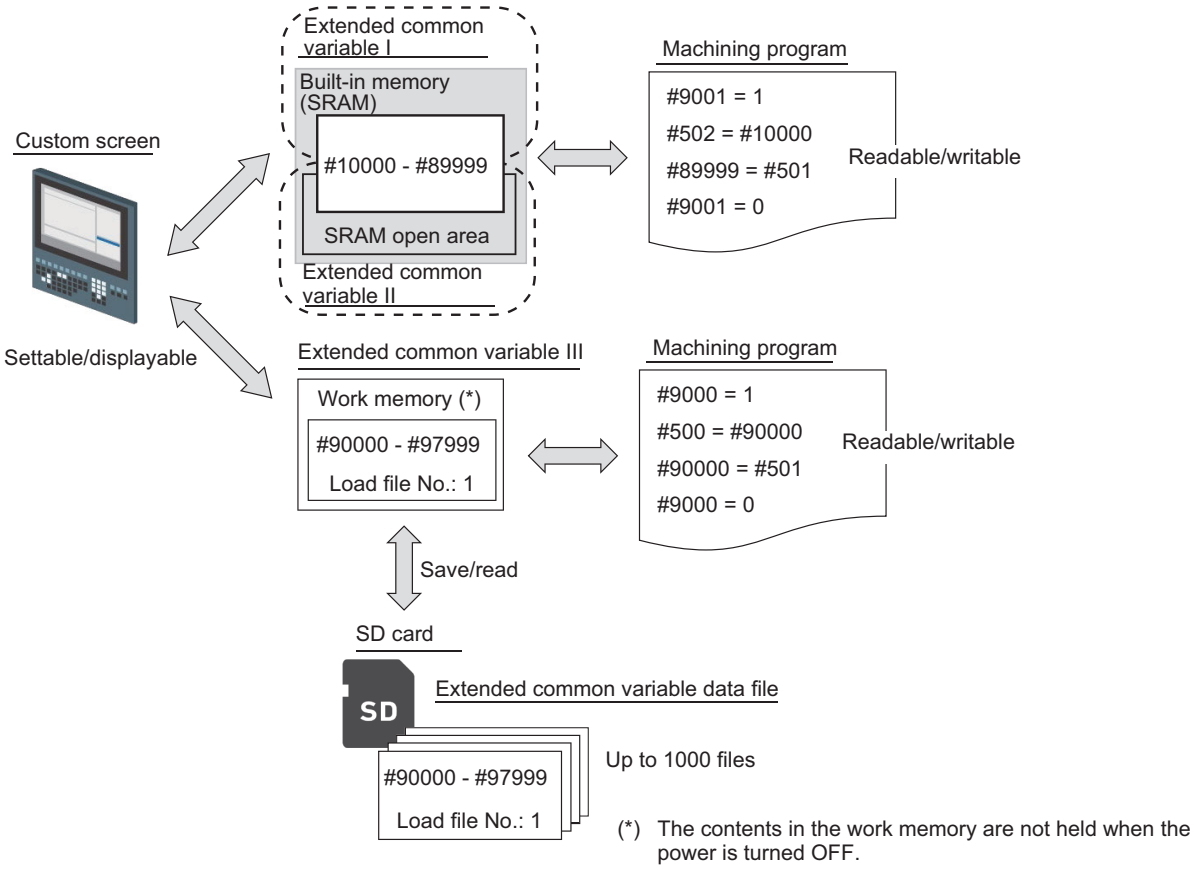
(1) Variables cannot be used with addresses "O" and "N".

12.1.2.5 Extended Common Variable

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

In addition to common variables, 80000 sets (#10000 to #89999) of extended common variable I/II and up to 8 million sets (8000 sets (#90000 to # 97999) × 1000 files) of extended common variable III can be used.
The extended common variable I/II data is saved in NC internal data (SRAM).
The extended common variable III data is stored as the extended common variable data file on an SD card.

Extended common variable can be used commonly for main program, subprogram and each macro program.
Command format, variable expressions, and variable quotations are the same as those of variable commands.



The number of sets of each variable can be set with the parameters.

[Extended common variable I]
There are variables that handle real values (up to 10000 sets) and variables that handle integer values (up to 40000 sets).
The total data size of real-type variables and integer-type variables is up to 80000 bytes.
Character strings can be set and displayed for the integer-type variables.
The variables are held even when the power is turned OFF.

[Extended common variable II]
A part of program storage area allocated with the SRAM open parameter is used as variables.
Only integer-type variables (up to 80000 sets) can be used.
The variables are held even when the power is turned OFF.

[Extended common variable III]
There are variables that handle real values (up to 2000 sets) and variables that handle integer values (up to 8000 sets).
Up to 8 million sets (8000 sets x 1000 files) of variables can be used by preparing multiple extended common variable data files.
The variables are read/written after the extended common variable data file is read out to the work memory.
The file can be saved/read out between the work memory and the SD card by the machining program (system variable #9000), custom API, PLC signals, etc.

Note

(1) Total number of extended common variable I and extended common variable II is up to 80000 sets.
(2) The upper limit of the extended common variable II data size is free space of SRAM open space.
(3) An SD card is required to use the extended common variable III. Use a dedicated SD card as the SD card.

12.1.3 Fixed Cycles

[List of fixed cycles]

Type of fixed cycle	M system	L system						
	G code list: 1	G code list: 2	G code list: 3	G code list: 4	G code list: 5	G code list: 6	G code list: 7	
Fixed cycle for drilling	G73	G80	G80	G80	G80	G80	G80	Refer to " 12.1.3.1 Fixed Cycle for Drilling". Refer to " 12.1.3.2 Fixed Cycle for Drilling (Type II)". Refer to " 4.5.3 Synchronous Tapping".
	:	:	:	:	:	:	:	
	G89	G89	G89	G89	G89	G89	G89	
		G79	G83.2	G79	G83.2	G79	G83.2	
	G98	G98	G98	G98	G98	G98	G98	
	G99	G99	G99	G99	G99	G99	G99	
	G187	G187	G187	G187	G187	G187	G187	
Special fixed cycle	G34							Refer to " 12.1.3.3 Special Fixed Cycle".
	G35	-	-	-	-	-	-	
	G36							
Fixed cycle for turning machining	G174	G90	G77	G90	G77	G90	G77	Refer to " 12.1.3.4 Fixed Cycle for Turning Machining".
	G175	G92	G78	G92	G78	G92	G78	
	G176	G94	G79	G94	G79	G94	G79	
Compound type fixed cycle for turning machining		G70	G70	G70	G70	G70	G70	Refer to " 12.1.3.5 Compound Type Fixed Cycle for Turning Machining". Refer to " 12.1.3.6 Compound Type Fixed Cycle for Turning Machining (Type II)". Refer to " 12.1.8.8 Multi-Part System Simultaneous Thread Cutting".
		:	:	:	:	:	:	
	-	G76	G76	G76	G76	G76	G76	
		G76.1	G76.1	G76.1	G76.1	G76.1	G76.1	
		G76.2	G76.2	G76.2	G76.2	G76.2	G76.2	
Small-diameter deep-hole drilling cycle	G83	-	-	-	-	-	-	

12.1.3.1 Fixed Cycle for Drilling

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

M system

These functions enable drilling, tapping and other hole machining cycles to be assigned in a simple 1-block program.

G code	Function	Retract at high speed	Chip removal (*1)	Cutting reduction amount specification (*2)
G80	Cancel	-	-	Cancel
G81	Drilling cycle Spot drilling cycle	Available	Available	-
G82	Drilling cycle Counter boring cycle	-	Available	-
G83	Deep-hole drilling cycle	Available	Available	Available
G84 (*3)	Tapping cycle	-	-	-
	Deep-hole tapping cycle Pecking tapping cycle	-	-	Available
G85/G86	Boring cycle	-	-	-
G87	Back boring cycle	-	-	-
G88/G89	Boring cycle	-	-	-
G73	Step cycle	Available	Available	Available
G74 (*3)	Reverse tapping cycle	-	-	-
	Deep-hole reverse tapping cycle Pecking reverse tapping cycle	-	-	Available
G75	Circular cutting cycle	-	-	-
G76	Fine boring cycle	-	-	-
G187	Thread milling cycle	-	-	-
G84.5/G84.6/G84.8 (*4)	Punchtap cycle	-	-	-
G74.5/G74.6/G74.8 (*4)	Reverse Punchtap cycle	-	-	-

(*1) Chips adhered to the tool can be removed by spindle reversal.
(*2) Hole drilling can be performed by specifying the cutting reduction amount while the depth of cut in each cutting operation (step) is reduced.
(*3) Which cycle is selected between tapping, deep-hole tapping, and pecking tapping depends on the presence or absence of the Q command and the setting of the parameter.
(*4) These cycles synchronously control the spindle and the servo axis using Punchtap-exclusive tools to cut into the work-piece and execute tapping.

A fixed cycle mode is canceled by G80 command or G command in the 01 group.

There are two levels of hole machining axis return which apply upon completion of the fixed cycle machining operation.

G code	Function
G98	Initial point level return
G99	R point level return

The basic program format for the fixed cycle commands is shown below.

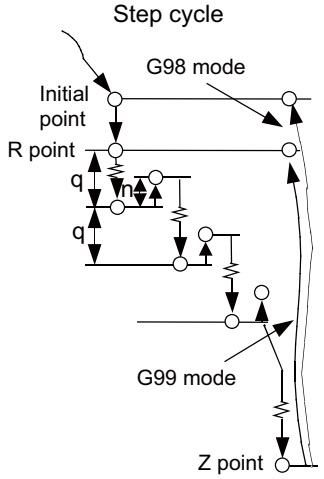
G8*(G7*) X__ Y__ Z__ R__ Q__ P__ F__ L__ S__ ,S__ ,I__ ,J__ D__ E__ J__ ,K__ ;

G8*(G7*) X__ Y__ Z__ R__ Q__ P__ F__ L__ S__ ,R__ ,I__ ,J__ D__ E__ J__ ,K__ ;

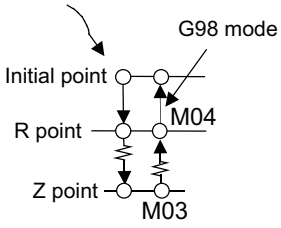
G8*(G7*)	Hole drilling mode
X/Y	Hole position data
Z/R/Q/P/F/J/,K	Hole machining data
L	Number of repetitions
S	Spindle rotation speed
,S	Spindle rotation speed during retract
,R	Synchronization switch
,I	Positioning axis in-position width
,J	Drilling axis in-position width
D	Spindle number for which chip removal is made (It can be commanded only in G81, G82, G83, and G73.)
E	Frequency in which chip removal is made (It can be commanded only in G81, G82, G83 and G73.)

For details on the synchronous tapping cycle (including pecking tapping cycle and deep-hole tapping cycle), refer to " 4.5.3 Synchronous Tapping".

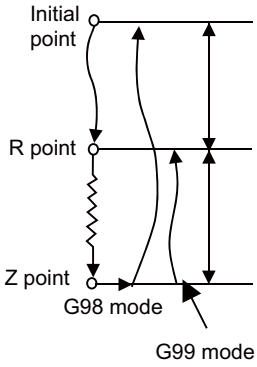
G73
Step cycle



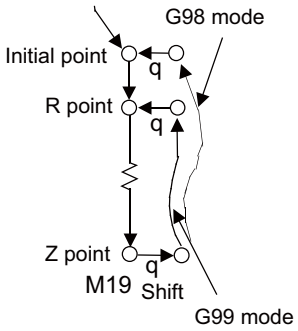
G74
Reverse tapping cycle



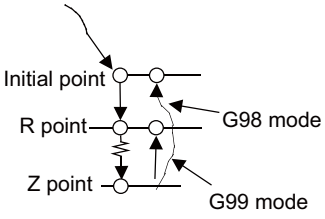
G75
Perfect circle cutting cycle



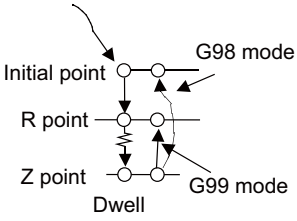
G76
Fine boring cycle



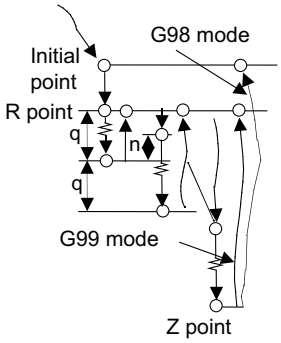
G81
Drilling, spot drilling cycle



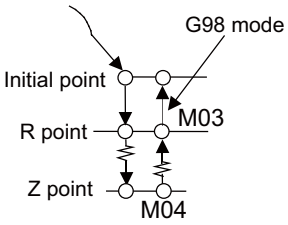
G82
Drilling, counterboring cycle



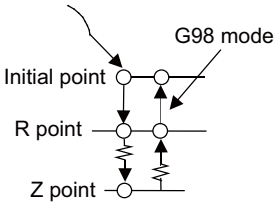
G83
Deep-hole drilling cycle



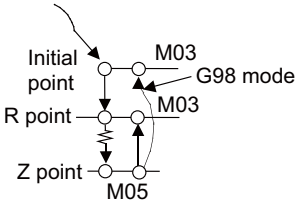
G84
Tapping cycle



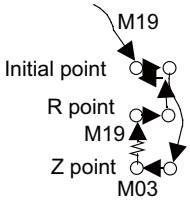
G85
Boring cycle



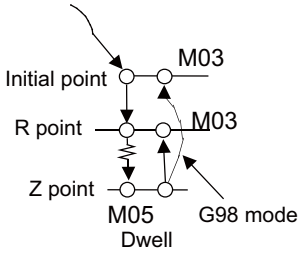
G86
Boring cycle



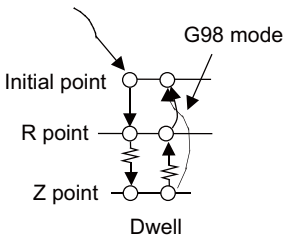
G87
Back boring cycle



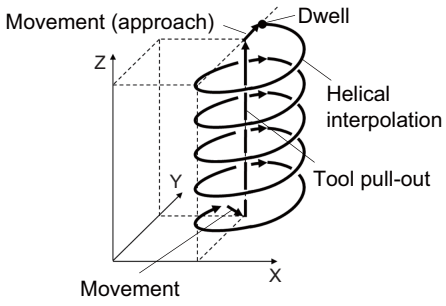
G88
Boring cycle



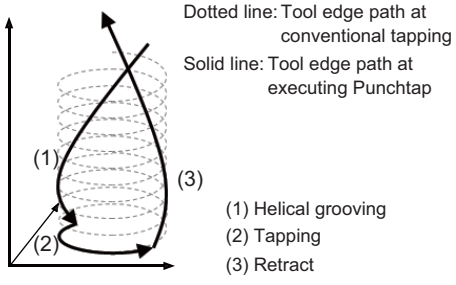
G89
Boring cycle



G187
Thread milling cycle



G84.5 (G84.6, G84.8)
Punchtap cycle



L system

These functions enable drilling, tapping and other hole machining cycles to be assigned in a simple 1-block program.

G code	Machining portion	Function	Retract at high speed	Chip removal (*1)	Cutting reduction amount specification (*2)
G80		Cancel	-	-	Cancel
G83 (*3)	Face	Deep-hole drilling cycle 1 High-speed deep-hole drilling cycle 1	Available	Available	Available
G83.5	Face	High-speed deep-hole drilling cycle 1	Available	Available	Available
G83.6	Face	Deep-hole drilling cycle 1	Available	Available	Available
G84 (*4)	Face	Tapping cycle Deep-hole tapping cycle Pecking tapping cycle	-	-	Available
G84.1 (*4)	Face	Reverse tapping cycle Deep-hole reverse tapping cycle Pecking reverse tapping cycle	-	-	Available
G85	Face	Boring cycle	-	-	-
G87 (*3)	Longitudinal	Deep-hole drilling cycle 1 High-speed deep-hole drilling cycle 1	Available	Available	Available
G87.5	Longitudinal	High-speed deep-hole drilling cycle 1	Available	Available	Available
G87.6	Longitudinal	Deep-hole drilling cycle 1	Available	Available	Available
G88 (*4)	Longitudinal	Tapping cycle Deep-hole tapping cycle Pecking tapping cycle	-	-	Available
G88.1 (*4)	Longitudinal	Reverse tapping cycle Deep-hole reverse tapping cycle Pecking reverse tapping cycle	-	-	Available
G89	Longitudinal	Boring cycle	-	-	-
G83.2	Face/Longitu-dinal	Deep-hole drilling cycle 2	Available	Available	-
G185		Hole edge chamfering cycle	-	-	-
G187		Thread milling cycle	-	-	-

(*1) Chips adhered to the tool can be removed by spindle reversal.

(*2) Hole drilling can be performed by specifying the cutting reduction amount while the depth of cut in each cutting operation (step) is reduced.

(*3) The difference between the deep-hole drilling cycle 1 and high-speed deep-hole drilling cycle 1 is the return amount after each step is completed.
Which cycle is used depends on the setting of the parameter.

(*4) Which cycle is selected between tapping, deep-hole tapping, and pecking tapping depends on the presence or absence of the Q command and the setting of the parameter.

A fixed cycle mode is canceled by G80 command or G command in the 01 group.

There are two levels of hole machining axis return which apply upon completion of the fixed cycle machining operation.

G code	Function
G98	Initial point level return
G99	R point level return

■ Command format

Face drilling

G8* X/U__ C/H__ Z/W__ R__ Q__ P__ F__ K__ D__ S__ ,S__ M__ E__ J__ ,K__ ;

G8*	Hole machining mode (G83*, G84*, G85)
X/U/C/H	Hole position data
Z/W/R/Q/P/F/J/,K	Hole machining data
K	Number of repetitions
D	Tool spindle number/Spindle number for which chip removal is made (It can be commanded only in G83* and G83.2.)
S	Spindle rotation speed
,S	Spindle rotation speed during retract
M	Miscellaneous command
E	Frequency in which chip removal is made (It can be commanded only in G83* and G83.2.)

Longitudinal drilling

G8* Z/W__ C/H__ X/U__ R__ Q__ P__ F__ K__ D__ S__ ,S__ M__ E__ J__ ,K__ ;

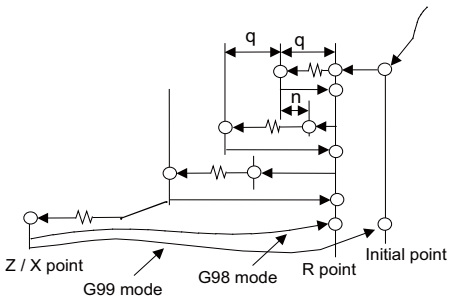
G8*	Hole machining mode (87*, G88*, G89)
Z/W/C/H	Hole position data
X/U/R/Q/P/F/J/,K	Hole machining data
K	Number of repetitions
D	Tool spindle number/Spindle number for which chip removal is made (It can be commanded only in G87* and G83.2.)
S	Spindle rotation speed
,S	Spindle rotation speed during retract
M	Miscellaneous command
E	Frequency in which chip removal is made (It can be commanded only in G87* and G83.2.)

Cancel

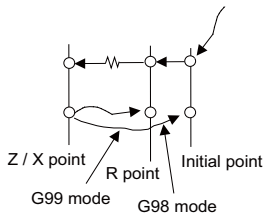
G80 ;

G83/G87
Deep-hole drilling cycle (G83: Z-axis direction, G87: X-axis direction)

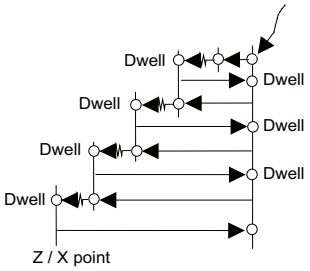
When Q command is given



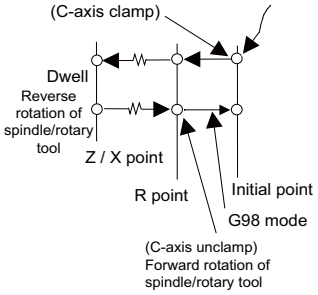
When Q command is not given



G83.2
Deep-hole drilling cycle

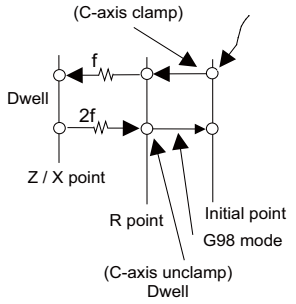


G84/88
Tapping cycle



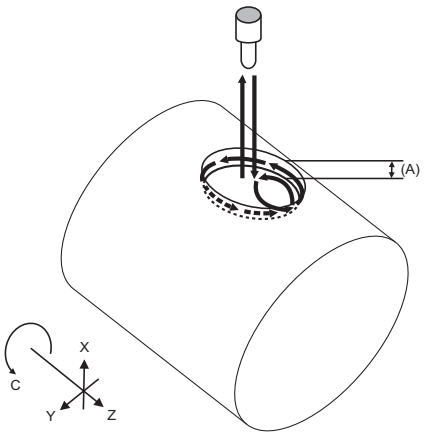
・ Output or no output can be set using a parameter for the C-axis clamp/unclamp M code
・ The rotation direction of the tapping is reversed using Reverse tapping cycle G84.1 (G88.1).

G85/89
Boring cycle



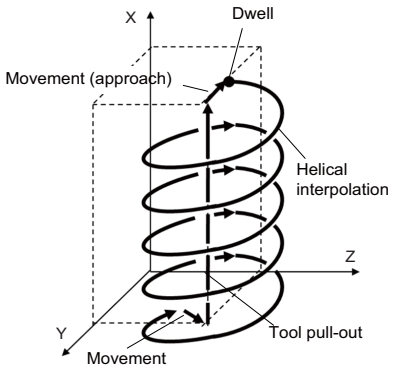
・ Output or no output can be set using a parameter for the C-axis clamp/unclamp M code

G185
Hole chamfering cycle



(A) Chamfering width

G187
Thread milling cycle



12.1.3.2 Fixed Cycle for Drilling (Type II)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	O	O	O	O	O	O	O

In the fixed cycle for drilling along the longitudinal direction, the X axis is specified as the drilling axis. However, in the fixed cycle for drilling (type II), the drilling axis can be changed to the Y axis by the PLC signal.

The applicable G codes are as follows:

G code	Function
G87 (*1)	Deep-hole drilling cycle 1 High-speed deep-hole drilling cycle 1
G87.5	High-speed deep-hole drilling cycle 1
G87.6	Deep-hole drilling cycle 1
G88 (*2)	Tapping cycle Deep-hole tapping cycle Pecking tapping cycle
G88.1 (*2)	Reverse tapping cycle Deep-hole reverse tapping cycle Pecking reverse tapping cycle
G89	Boring cycle

(*1) The difference between the deep-hole drilling cycle 1 and high-speed deep-hole drilling cycle 1 is the return amount after each step is completed.
Which cycle is used depends on the setting of the parameter.

(*2) Which cycle is selected between tapping, deep-hole tapping, and pecking tapping depends on the presence or absence of the Q command and the setting of the parameter.

12.1.3.3 Special Fixed Cycle

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	O	O	O
L	—	—	—	—	—	—	—

These functions enable drilling, tapping and other hole machining cycles to be assigned in a simple 1-block program. Special fixed cycles must always be used in combination with fixed cycles.

The special fixed cycles are as follows:

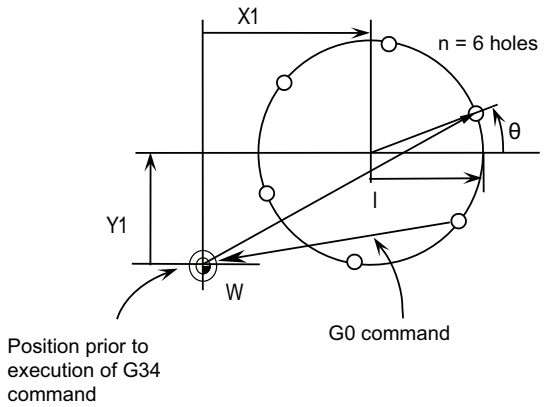
G code	Function
G34	Bolt hole circle
G35	Line at angle
G36	Arc
G37.1	Grid

Bolt hole circle (G34)

The tool starts at the point forming angle θ with the X axis on the circumference of a circle with radius R whose center is the coordinates designated by X and Y, and it drills "n" number of holes at "n" equal intervals along the circumference of that circle. The drilling data for the standard fixed cycle of the G81 or other such command is retained for the drilling operation at each hole position.
All movements between the hole positions are conducted in the G00 mode. The data is not retained upon completion of the G34 command.

G34 Xx Yy Ir Jθ Kn ;

Xx,Yy	Center position of bolt hole circle They are affected by the G90/G91 commands.
Ir	Radius "r" of circle It is based on the least command increment and is provided with a positive number.
Jθ	Angle θ at the point to be drilled initially The counterclockwise direction is taken to be positive.
Kn	Number "n" of holes to be drilled Any number of holes from 1 through 9999 can be designated; 0 cannot be assigned. When 0 has been designated, the alarm will occur. A positive number provides positioning in the counterclockwise direction; a negative number provides positioning in the clockwise direction.



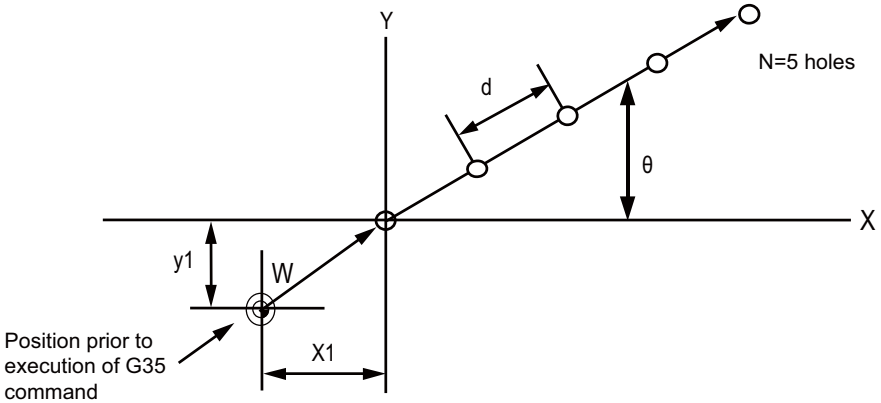
As shown in the figure, the tool is positioned above the final hole upon completion of the G34 command. This means that when it is to be moved to the next position, if the command is issued with incremental position, it will be necessary to calculate the coordinate position. Thus, it is convenient to issue the command with absolute position.

Line at angle (G35)

With the starting point at the position designated by X and Y, the tool drills "n" number of holes each at interval "d" in the direction forming angle θ with the X axis. A standard fixed cycle applies for the drilling operation at each of the hole positions and so there is a need to retain beforehand the drilling data (drilling mode and drilling data). All movements between the hole positions are conducted in the G00 mode. The data is not retained upon completion of the G35 command.

G35 Xx Yy Id Jθ Kn ;

Xx,Yy	The starting point coordinates They are affected by the G90/G91 commands.
Id	Interval "d" It is based on the least command increment and when "d" is negative, drilling proceeds in the point symmetrical direction centered on the starting point.
Jθ	Angle θ The counterclockwise direction is taken to be positive.
Kn	Number "n" of holes to be drilled Number "n" of holes to be drilled including the starting point; any number of holes from 1 through 9999 can be assigned.

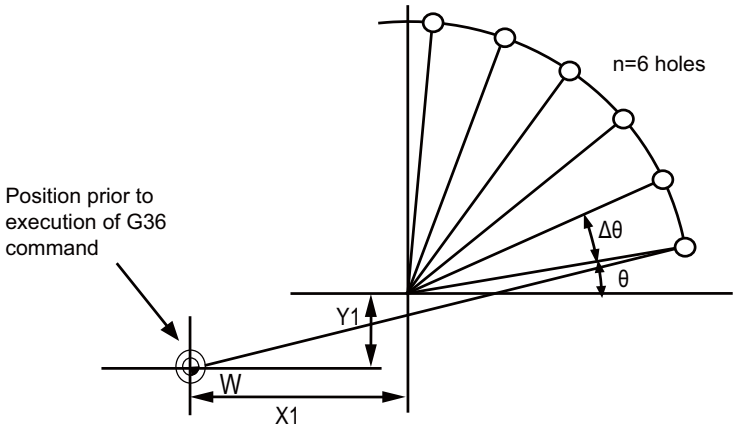


Arc (G36)

The tool starts at the point forming angle θ with the X axis on the circumference of a circle with radius "r" whose center is the coordinates designated by X and Y, and it drills "n" number of holes aligned at angle interval Δθ. As with the bolt hole circle function, the drilling operation at each of the hole positions is based on a hold drilling fixed cycle and so there is a need to retain the drilling data beforehand.
All movements between the hole positions are conducted in the G00 mode. The data is not retained upon completion of the G36 command.

G36 Xx Yy Ir Jθ PΔθ Kn ;

Xx,Yy	Center coordinates of arc They are affected by the G90/G91 commands.
Ir	Radius "r" of arc It is based on the least command increment and is provided with a positive number.
Jθ	Angle θ at the point to be drilled initially The counterclockwise direction is taken to be positive.
P Δθ	Angle interval Δθ When it is positive, the tool drills in the counterclockwise direction and when it is negative, it drills in the clockwise direction.
Kn	Number "n" of holes to be drilled Any number of holes from 1 through 9999 can be assigned.

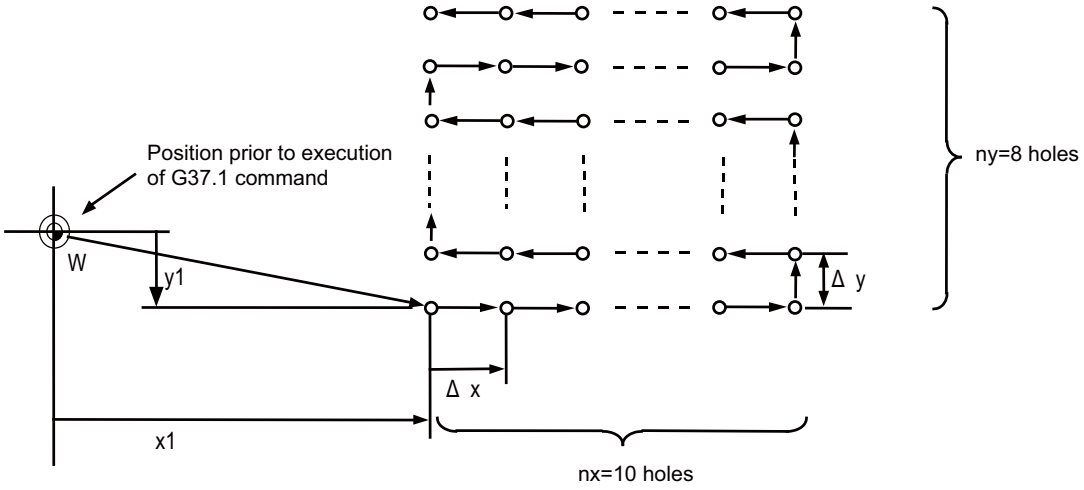


Grid (G37.1)

With the starting point at on the position designated by X and Y, this function enables the tool to drill the holes on the lattice with "nx" number of holes at parallel intervals of Δx to the X axis. Drilling proceeds in the X-axis direction. The drilling operation at each of the hole positions is based on a standard fixed cycle and so there is a need to command the drilling data (drilling mode and drilling data) beforehand. All movements between the hole positions are conducted in the G00 mode. The data is not retained upon completion of the G37.1 command.

G37.1 Xx Yy IΔx Pnx JΔy Kny ;

Xx,Yy	The starting point coordinates They are affected by the G90/G91 commands.
I Δx	X-axis interval Δx It is based on the least command increment; when Δx is positive, the intervals are provided in the positive direction as seen from the starting point and when it is negative, they are provided in the negative direction.
Pnx	Number of holes "nx" in the X-axis direction The specifiable range is 1 to 9999.
J Δy	Y-axis interval Δy It is based on the least command increment; when Δy is positive, the intervals are provided in the positive direction as seen from the starting point and when it is negative, they are provided in the negative direction.
Kny	Number of holes "ny" in the Y-axis direction The specifiable range is 1 to 9999.



12.1.3.4 Fixed Cycle for Turning Machining

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	—
L	O	O	O	O	O	O	O

By using this function, the shape normally programmed in several blocks for rough cutting, etc., in the turning machining can be commanded in one block.
By editing the standard fixed cycle subprograms, the fixed cycle sequences can be changed by the user. The user can also register and edit an original fixed cycle program.

G code [L system]	G code [M system]	Function
G77	G174	Longitudinal cutting fixed cycle
G78	G175	Thread cutting fixed cycle
G79	G176	Face cutting fixed cycle

Longitudinal cutting fixed cycle (G77/G174)

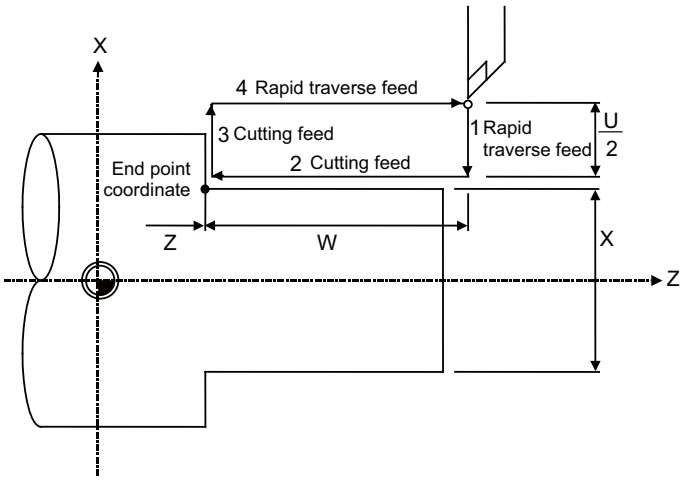
■ Straight cutting

Straight cutting in the longitudinal direction can be performed consecutively by the following block:

G77 X/U__ Z/W__ F__ ;	**[L system]**

G174 X__ Z__ F__ ;	**[M system]**

X/U	X axis end point coordinate
Z/W	Z axis end point coordinate
F	Feedrate



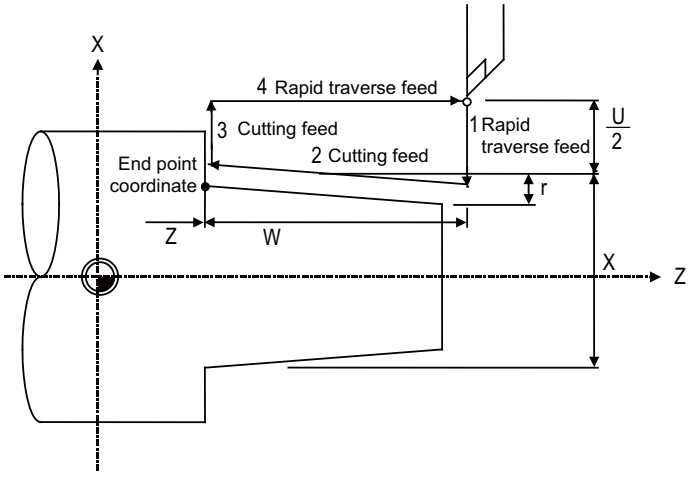
■ Taper cutting

Taper cutting in the longitudinal direction can be performed consecutively by the following block:

G77 X/U__ Z/W__ R__ F__ ;	**[L system]**

G174 X__ Z__ R__ F__ ;	**[M system]**

X/U	X axis end point coordinate
Z/W	Z axis end point coordinate
R	Taper part depth (radius designation, incremental position, sign is required)
F	Feedrate



r: Taper part depth (radius designation, incremental position, sign is required)

Thread cutting fixed cycle (G78/G175)

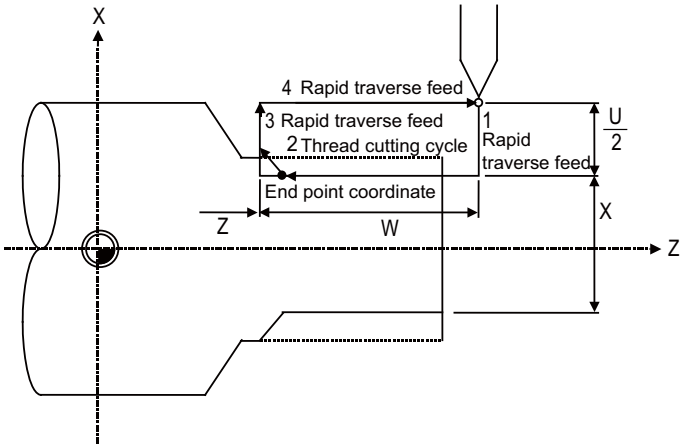
■ Straight thread cutting

Straight thread cutting can be performed by the following block:

G78 X/U__ Z/W__ F/E__ Q__ ; [L system]

G175 X__ Z__ F/E__ Q__ ; [M system]

X/U	X axis end point coordinate
Z/W	Z axis end point coordinate
F/E	Lead of Longitudinal axis (axis which moves most) direction
Q	Thread cutting start shift angle



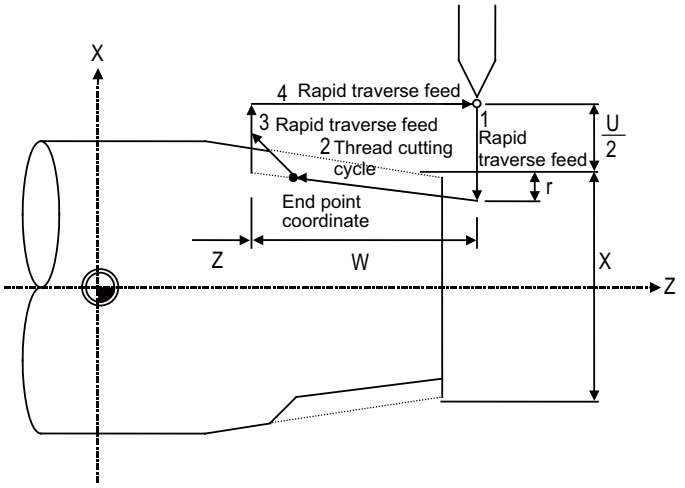
■ Taper thread cutting

Taper thread cutting can be performed by the following block:

G78 X/U__ Z/W__ R__ F/E__ Q__ ; [L system]

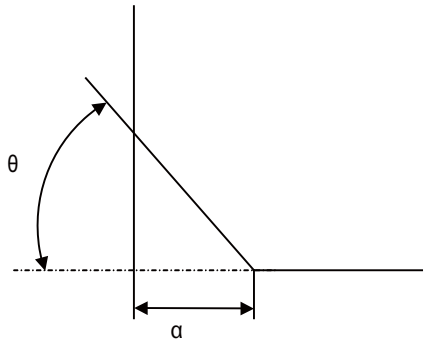
G175 X__ Z__ R__ F/E__ Q__ ; [M system]

X/U	X axis end point coordinate
Z/W	Z axis end point coordinate
R	Taper part depth (radius designation, incremental position, sign is required)
F/E	Lead of Longitudinal axis (axis which moves most) direction
Q	Thread cutting start shift angle



r: Taper part depth (radius designation, incremental position, sign is required)

[Chamfering]



α: Thread chamfering amount

Set the chamfer amount with the parameters.

The available range is 0 to 12.7 leads. It can be set with increments of 0.1L

θ: Thread chamfering angle

The thread chamfering angle can be set in a given parameter in 1° steps in the range of 0 to 89°

Face cutting fixed cycle (G79/G176)

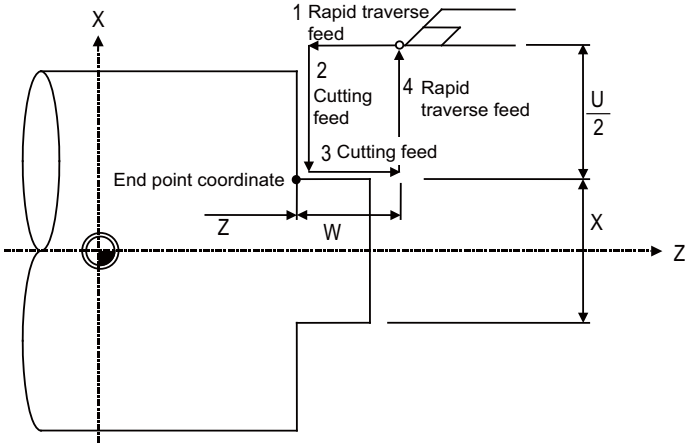
■ Straight cutting

Straight cutting in the end face direction can be performed consecutively by the following block:

G79 X/U__ Z/W__ F__ ;	**[L system]**

G176 X__ Z__ F__ ;	**[M system]**

X/U	X axis end point coordinate
Z/W	Z axis end point coordinate
F	Feedrate



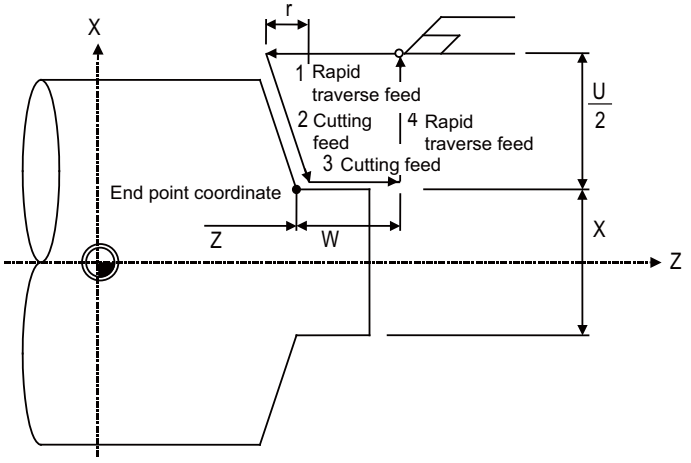
■ Taper cutting

Taper cutting in the end face direction can be performed consecutively by the following block:

G79 X/U__ Z/W__ R__ F__ ;	**[L system]**

G176 X__ Z__ R__ F__ ;	**[M system]**

X/U	X axis end point coordinate
Z/W	Z axis end point coordinate
R	Taper part depth (radius designation, incremental position, sign is required)
F	Feedrate



r: Taper part depth (radius designation, incremental position, sign is required)

12.1.3.5 Compound Type Fixed Cycle for Turning Machining

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	Δ	Δ	Δ	Δ	○	○	○

The shape normally programmed in several blocks for rough cutting, etc., in the turning machining can be commanded in one block. This function is useful for machining program simplification.
Compound type fixed cycle for turning machining are as follows:

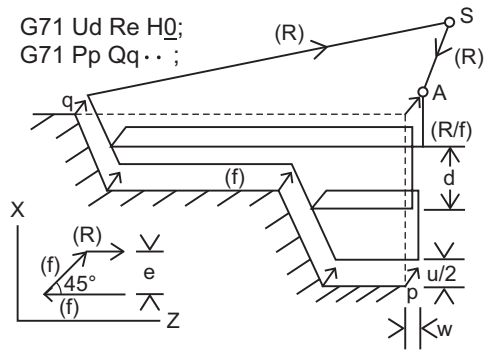
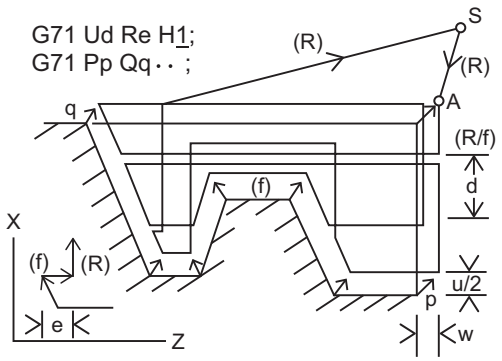
G code	Function	
G71	Longitudinal rough cutting cycle	Compound type fixed cycle for turning machining I
G72	Face rough cutting cycle	
G73	Formed material rough cutting cycle	
G70	Finish cycle	
G74	Face cut-off cycle	Compound type fixed cycle for turning machining II
G75	Longitudinal cut-off cycle	
G76	Compound type thread cutting cycle	

Longitudinal rough cutting cycle (G71)

The finish shape program is called, and straight rough cutting is performed while intermediate path is being calculated automatically.

G71 Ud Re Hh ;
G71 Aa Pp Qq Uu Ww Ff Ss Tt ;

Ud	Cut depth (modal) ..... Can also be set with the Reversible parameter. Increment: μm or 1/10000 inch (Radius value command)
Re	Retract amount (modal) ... Can also be set with the Reversible parameter. Increment: μm or 1/10000 inch (Radius value command)
Hh	Pocket machining (modal) ..... Can also be set with the Reversible parameter. 0: No pocket machining 1: Pocket machining
Aa	Finished shape program No. (Program being executed when omitted) A four-digit or eight-digit program No. starting with O No. can specify the finish shape program by the parameter.
Pp	Finished shape start sequence No. (program head when omitted)
Qq	Finished shape end sequence No. (To the end of the program when omitted) However, if M99 precedes the Q command, up to M99.
Uu	Finishing allowance in X axis direction (If omitted, finishing allowance in X axis direction is handled as "0".) Increment: μm or 1/10000 inch (Whether diameter value or radius value is commanded depends on the parameter setting.)
Ww	Finishing allowance in Z axis direction (If omitted, finishing allowance in Z axis direction is handled as "0".) Increment: μm or 1/10000 inch (Radius value command)
Ff	Cutting feedrate (If omitted, cutting feedrate (modal) before G71 is applied.)
Ss, Tt	Spindle command, tool command

Example of a finished shape without pocket	Example of a finished shape with pocket
	
With the beginning of the pockets, the tool is pulled up in the 45-degree direction with each cycle, and finally roughing along finishing shape is performed. By setting parameters, the tool can be moved to the cycle command point at rapid traverse without roughing along finishing shape.	A method that traces the finished shape with each cycle is used for the beginning of the pockets. Roughing along finishing shape can be performed by setting parameters.
S: Cycle command point A: Rough cutting start point (R) Rapid traverse (f) Cutting feed	

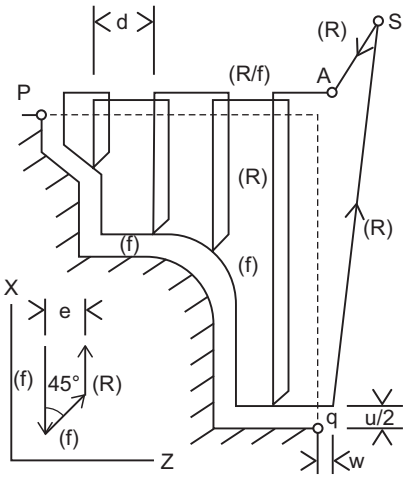
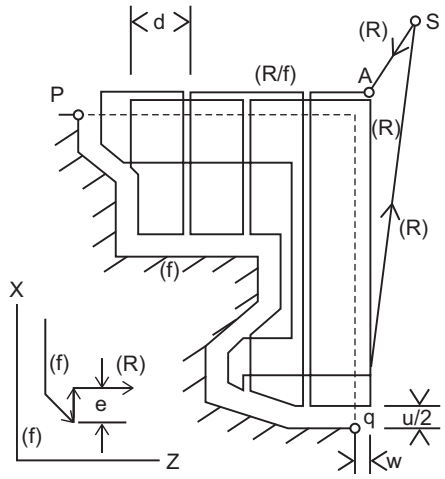
The path to the cycle command point after roughing along finishing shape and whether or not to retract a tool after roughing along finishing shape can be selected by parameters.

Face rough cutting cycle (G72)

The finish shape program is called, and rough cutting is performed in the end face direction while intermediate path is being calculated automatically.

G72 Wd Re Hh ;
G72 Aa Pp Qq Uu Ww Ff Ss Tt ;

Wd	Cut depth (modal) ..... Can also be set with the Reversible parameter. Unit: μm/1/10000 inch (Radius value command)
Re	Retract amount (modal) ... Can also be set with the Reversible parameter. Unit: μm/1/10000 inch (Radius value command)
Hh	Pocket machining (modal) ..... Can also be set with the Reversible parameter. 0: No pocket machining 1: Pocket machining
Aa	Finished shape program No. (Program being executed when omitted) A four-digit or eight-digit program No. starting with O No. can specify the finish shape program by the parameter.
Pp	Finished shape start sequence No. (program head when omitted)
Qq	Finished shape end sequence No. (To the end of the program when omitted) However, if M99 precedes the Q command, up to M99.
Uu	Finishing allowance in X axis direction (If omitted, finishing allowance in X axis direction is handled as "0".) Increment: μm or 1/10000 inch (Whether diameter value or radius value is commanded depends on the parameter setting.)
Ww	Finishing allowance in Z axis direction (If omitted, finishing allowance in Z axis direction is handled as "0".) Increment: μm or 1/10000 inch (Radius value command)
Ff	Cutting feedrate (If omitted, cutting feedrate (modal) before G72 is applied.)
Ss, Tt	Spindle command, tool command

Example of a finished shape without pocket	Example of a finished shape with pocket
G72 Wd Re H0; G72 Pp Qq‥ ; 	G72 Wd Re H1; G72 Pp Qq‥ ; 
With the beginning of the pockets, the tool is pulled up in the 45-degree direction with each cycle, and finally roughing along finishing shape is performed. By setting parameters, the tool can be moved to the cycle command point at rapid traverse without roughing along finishing shape.	A method that traces the finished shape with each cycle is used for the beginning of the pockets. Roughing along finishing shape can be executed by setting parameters.
S: Cycle command point A: Rough cutting start point (R) Rapid traverse (f) Cutting feed	

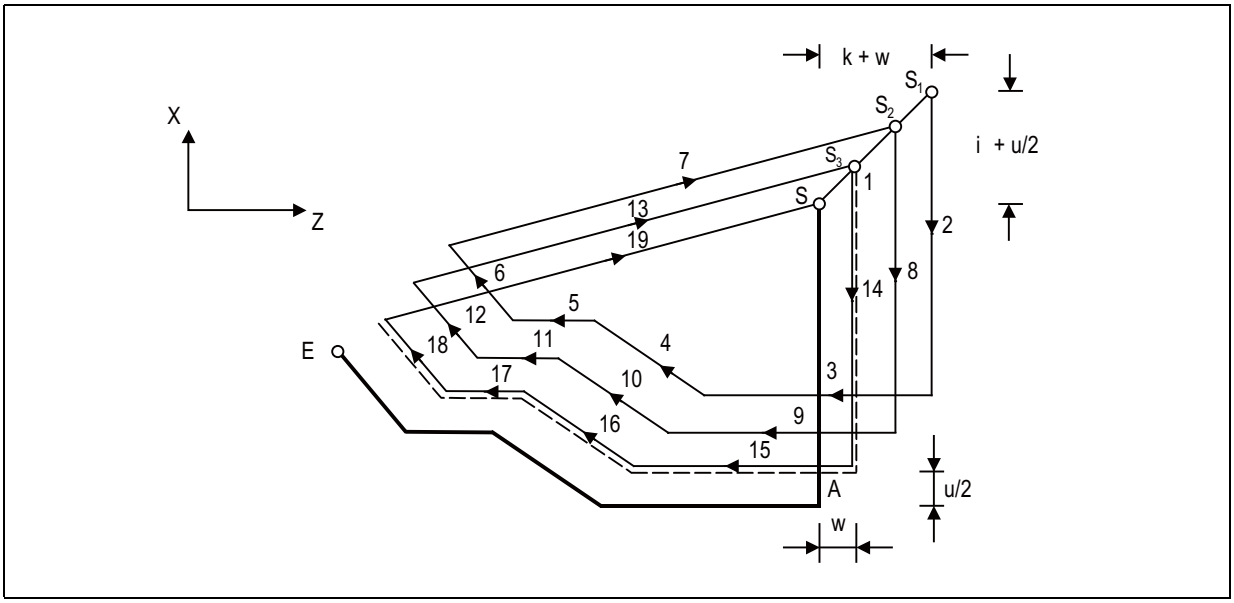
The path to the cycle command point after roughing along finishing shape and whether or not to retract a tool after roughing along finishing shape can be selected by parameters.

Formed material rough cutting cycle (G73)

The finish shape program is called. Intermediate path is automatically calculated and rough cutting is performed conforming to the finish shape.

G73 Ui Wk Rd ;
G73 Aa Pp Qq Uu Ww Ff Ss Tt ;

Ui	Cutting allowance in the X axis direction "i"	• Cutting allowance when P, Q command is not given. • Modal data • Sign is ignored. • Cutting allowance is given with a radius designation.
Wk	Cutting allowance in the Z axis direction "k"	
Rd	Split count "d"	
Aa	Finish shape program No.	(If it is omitted, the program being executed is assumed to be designated.) A file name can be designated instead of address A by enclosing the file name in angle brackets < >. (The file name can have up to 32 characters, including the extension.)
Pp	Finished shape start sequence No.	(If it is omitted, the program top is assumed to be designated.)
Qq	Finished shape end sequence No.	(If it is omitted, the program end is assumed to be designated.) However, if M99 precedes the Q command, up to M99.
Uu	Finishing allowance in the X axis direction "u"	• Finishing allowance for the finish shape commanded by the address P or Q. • Sign is ignored. • Diameter or radius is designated according to the parameter. • The shift direction is determined by the shape
Ww	Finishing allowance in the Z axis direction "w"	
Ff	Cutting feedrate (F function)	The F, S, and T commands in the finish shape program are ignored, and the value in the rough cutting command or the preceding value becomes effective.
Ss	Spindle speed (S function)	
Tt	Tool command (T function)	



Finish cycle (G70)

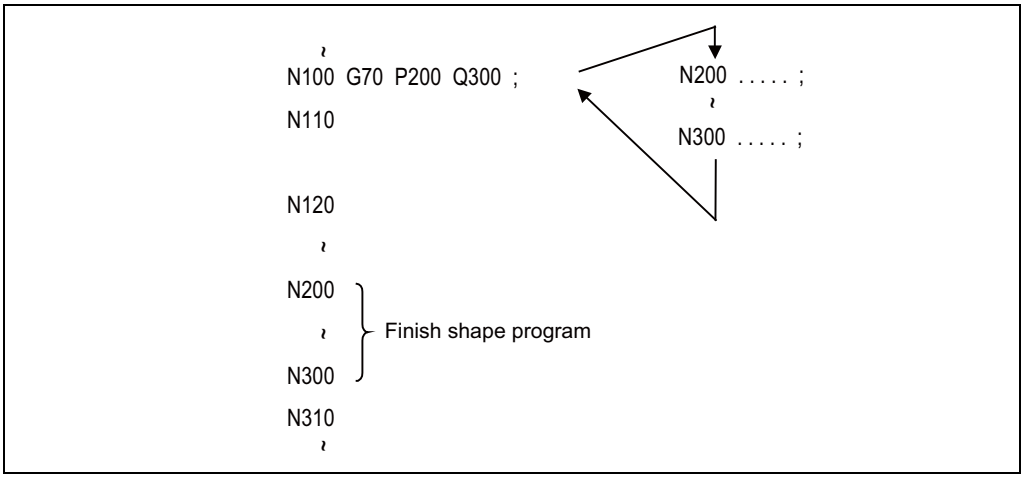
After rough cutting is performed by using G71 to G73, finish cutting can be performed by using the G70 command.

G70 Aa Pp Qq Hh;

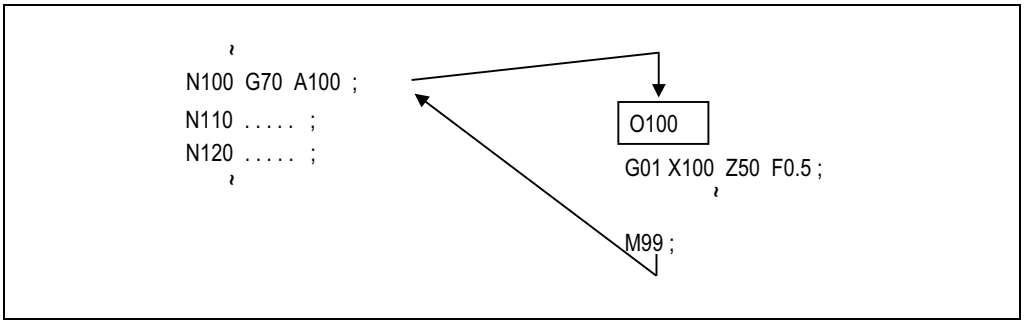
Aa	Finished shape program No. (Program being executed when omitted) A file name can be designated instead of address A by enclosing the file name in angle brackets < >. (The file name can have up to 32 characters, including the extension.)
Pp	Finished shape start sequence No. (program head when omitted)
Qq	Finished shape end sequence No. (To the end of the program when omitted) However, if M99 precedes the Q command, up to M99.
Hh	Tool feed directions when the cutting-edge angle is compensated 0: Not compensate the cutting-edge angle 1: Compensate the cutting-edge angle in the longitudinal direction (same tool feed direction as G71/G73 command) 2: Compensate the cutting-edge angle in the end face direction (same tool feed direction as G72 command) • When H address is omitted, the cutting-edge is not compensated. • When the tool shape compensation for turning is disabled, H address is ignored.

- The F, S, and T commands in the rough cutting cycle command G71 to G73 blocks are ignored, and the F, S, and T commands in the finish shape program become effective.
- The memory address of the finish shape program executed by G71 to G72 is not stored.
  Whenever G70 is executed, a program search is made.
- When the G70 cycle terminates, the tool returns to the start point at the rapid traverse feed rate and the next block is read.

[(Example 1) When a sequence No. is designated]



[(Example 2) When a program No. is designated]



In either example 1 or 2, after the N100 cycle is executed, the N110 block is executed.

12.1.3.6 Compound Type Fixed Cycle for Turning Machining (Type II)

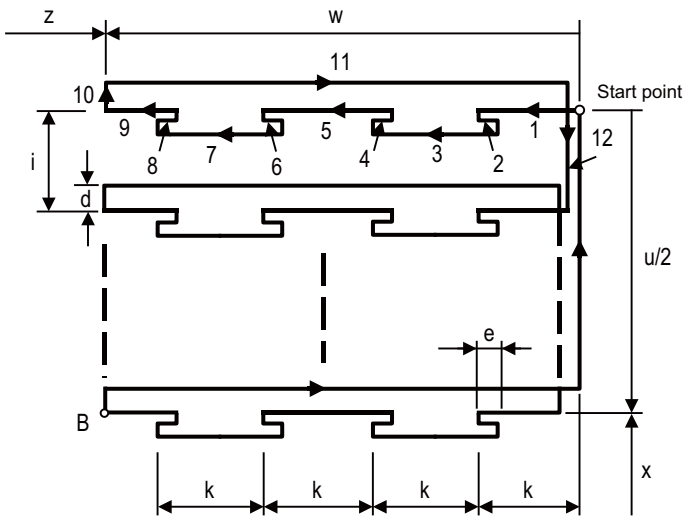
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	Δ	Δ	Δ	Δ	○	○	○

Face cutting-off cycle (G74)

When the slotting end point coordinates, cutting depth, cutting tool shift amount, and cutting tool relief amount at the cut bottom are commanded, automatic slotting is performed in the end face direction of a given bar by G74 fixed cycle.

G74 Re ;
G74 Xx1/(Uu1) Zz1/(Ww1) Pi Qk Rd Ff ;

Re	Retract amount "e" (when X/U, Z/W command is not given) (Modal)
Xx1/Uu1	B point coordinate (absolute/incremental position)
Zz1/Ww1	B point coordinate (absolute/incremental position)
Pi	Tool shift amount (radius designation, incremental position, sign not required)
Qk	Cut depth "k" (radius designation, incremental position, sign not required)
Rd	Escape amount "d" at cut bottom (If sign is not provided, relief is made at the first cut bottom. If minus sign is provided, relief is made not at the first cut bottom but made at the second cut bottom and later.)
Ff	Feedrate



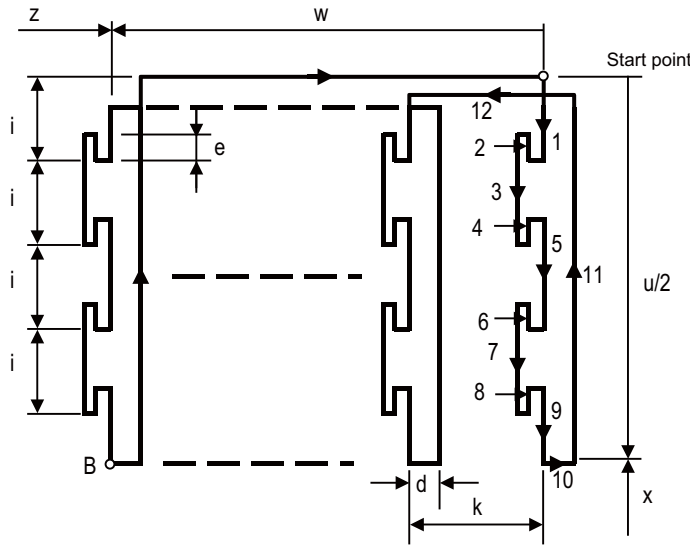
9 and 12 just before the last cycle are executed with the remaining distance. 2, 4, 6, 8, 10, 11 and 12 are executed at the rapid traverse feed rate.

Longitudinal cutting-off cycle (G75)

When the slotting end point coordinates, cutting depth, cutting tool shift amount, and cutting tool relief amount at the cut bottom are commanded, automatic slotting is performed in the longitudinal direction of a given bar by G75 fixed cycle.

```
G75 Re ;
G75 Xx1/(Uu1) Zz1/(Ww1) Pi Qk Rd Ff ;
```

Re	Retract amount "e" (when X/U, Z/W command is not given) (Modal)
Xx1/Uu1	B point coordinate (absolute/incremental position)
Zz1/Ww1	B point coordinate (absolute/incremental position)
Pi	Cut depth "i" (radius designation, incremental position, sign not required)
Qk	Tool shift amount "k" (radius designation, incremental position, sign not required)
Rd	Escape amount "d" at cut bottom (If sign is not provided, relief is made at the first cut bottom. If minus sign is provided, relief is made not at the first cut bottom but made at the second cut bottom and later.)
Ff	Feedrate



9 and 12 just before the last cycle are executed with the remaining distance. 2, 4, 6, 8, 10, 11 and 12 are executed at the rapid traverse feed rate.

Compound type thread cutting cycle (G76)

When the thread cutting start and end points are commanded, cutting at any desired angle can be made by automatic cutting so that the cutting cross section area (cutting torque) per time becomes constant in the G76 fixed cycle.
Various longitudinal threads can be cut by considering the thread cutting end point coordinate and taper height constituent command value.

■ Command format

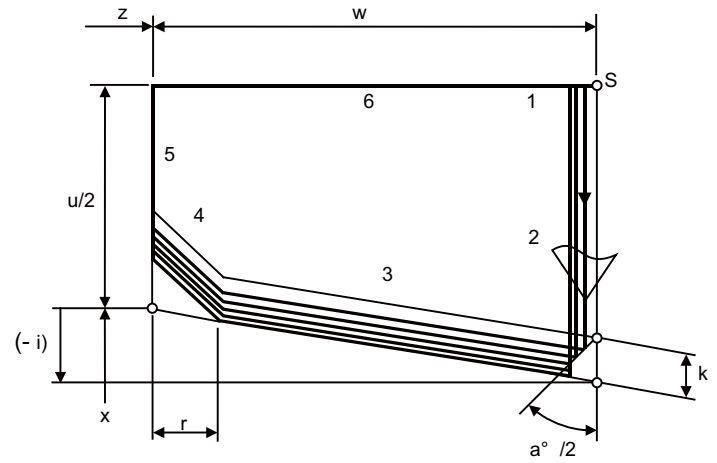
```
G76  Pmra QΔdmin Rd ;
G76  Xx1/Uu1  Zz1/Ww1  Ri  Pk  QΔd  Fl ;
```

m	Cutting count at finishing (01 to 99) (modal)
r	Chamfering amount (00 to 99) (modal) Set in 0.1-lead increments.
a	Tool nose angle (thread angle) (00 to 99) (modal) Set in 1-degree increments.
Δdmin	Minimum cut amount When the calculated cutting depth becomes smaller than Δdmin, the cutting depth is clamped at Δdmin.
d	Finishing allowance (modal)
Xx1/Uu1	X-axis end point coordinates of thread part Absolute/Incremental position
Zz1/Ww1	Z-axis end point coordinates of thread section Absolute/Incremental position
Ri	Taper height component (radius value) at thread section When "i" is "0", straight screw is made.
Pk	Thread height Designate the thread height in a positive radius value.
QΔd	Cut depth Designate the first cutting depth in a positive radius value.
Fl	Thread lead

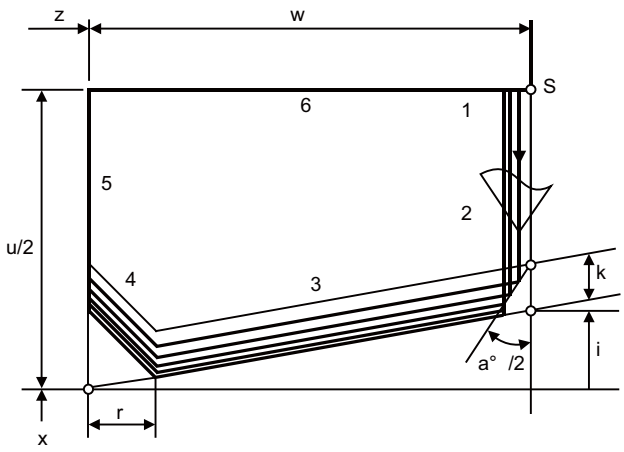
■ 1 cycle configuration

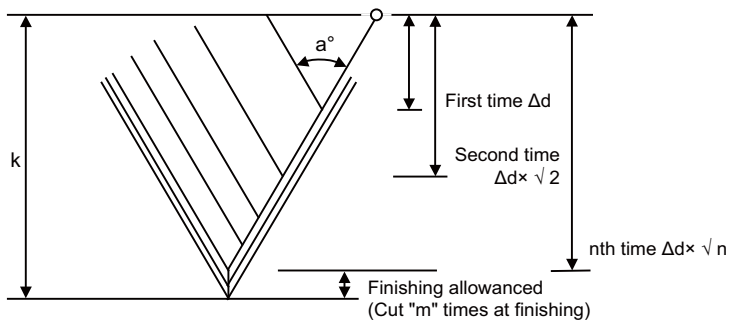
In one cycle, 1, 2, 5, and 6 move at rapid traverse feed and 3 and 4 move at cutting feed designated in F command.

When "Ri" is negative



When "Ri" is positive





12.1.3.7 Small-Diameter Deep-Hole Drilling Cycle

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	Δ	Δ	Δ	Δ	—	—	—

In deep-hole drilling, cutting and retract are repeated and the workpiece is machined multiple times. In addition, when PLC signals are input during cutting, the cutting for the time concerned is skipped. In this way, this cycle reduces the load applied to the tool.

■ Command format

The small-diameter deep-hole drilling cycle mode is established by designating the M code command which was set in the parameter.
If the G83 command is designated in this mode, the small-diameter deep-hole drilling cycle is executed.
The mode is canceled by the following conditions.

- Designation of a fixed cycle cancel command (G80, G commands in Group 1)
- Reset

The signal is not output in respect to the M command which changes the mode to the small-diameter deep-hole drilling mode.

G83 Xx1 Yy1 Zz1 Rr1 Qq1 Ff1 Ii1 Pp1 ;

Xx1, Yy1	Hole drilling position
Zz1	Hole bottom position
Rr1	R point position
Qq1	Depth of cutting in each pass (designated with incremental position)
Ff1	Cutting feedrate
Ii1	Feedrate from R point to the cutting start position, the speed (mm/min) for returning from hole bottom
Pp1	Dwell time at hole bottom position

12.1.3.8 Two-Dimensional Barcode Engraving Cycle

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

This function is used to machine a QR code that stores an arbitrary character string on the machined surface.
When the character string to be stored in the QR code is specified, the QR code that stores the character string can be machined on the cylindrical surface or plane.
In addition to the character string, the size of the QR code can be specified by commanding a specific address.

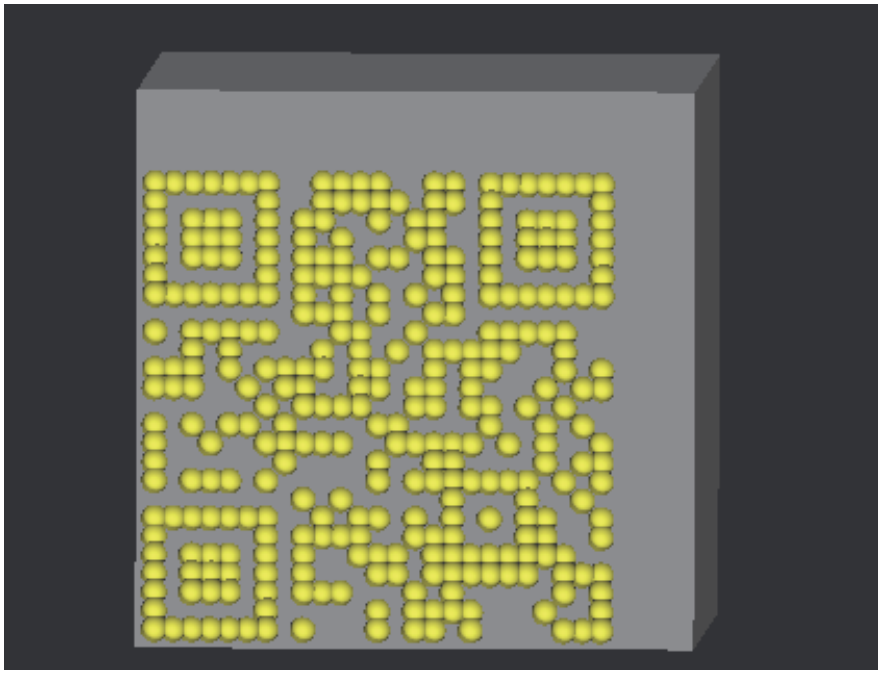
The QR code consists of a combination of white and black squares called cells.
In this function, the QR code is expressed as a collection of dots by performing a drilling operation in the black cell part.
Therefore, the size of the cell is determined by the tool diameter and depth.

■ Command format

G136 X_ Y_ Z_ α_ R_ D_ F_ L_ I_ P_ K_ E_ [_] ;

X, Y, Z, α	Command position
R	Evacuation height
D	Depth
F	Feedrate in cutting feed
L	QR code length
I	Cell interval
P	QR code position (0: Top right, 1: Bottom right, 2: Top left, 3: Bottom left, 4: Center)
K	QR code angle
E	Error correction level (0: 7%, 1: 15%, 2: 25%, 3: 30%)
[ ]	The character string to be stored in the QR code (maximum 100 characters)

■ Example of machining result



■ Open source software

The QR Code generator library, which is the MIT license, is used for this function.
The copyright notice and permission notice of the QR Code generator library are described below.

QR Code generator library
Copyright (c) 2019 Project Nayuki
Released under the MIT license

Permission is hereby granted, free of charge, to any person obtaining a copy of this software and associated documentation files (the "Software"), to deal in the Software without restriction, including without limitation the rights to use, copy, modify, merge, publish, distribute, sublicense, and/or sell copies of the Software, and to permit persons to whom the Software is furnished to do so, subject to the following conditions:

The above copyright notice and this permission notice shall be included in all copies or substantial portions of the Software.

THE SOFTWARE IS PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO THE WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NONINFRINGEMENT. IN NO EVENT SHALL THE AUTHORS OR COPYRIGHT HOLDERS BE LIABLE FOR ANY CLAIM, DAMAGES OR OTHER LIABILITY, WHETHER -26- IN AN ACTION OF CONTRACT, TORT OR OTHERWISE, ARISING FROM, OUT OF OR IN CONNECTION WITH THE SOFTWARE OR THE USE OR OTHER DEALINGS IN THE SOFTWARE.

12.1.3.9 Text Engraving Cycle

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

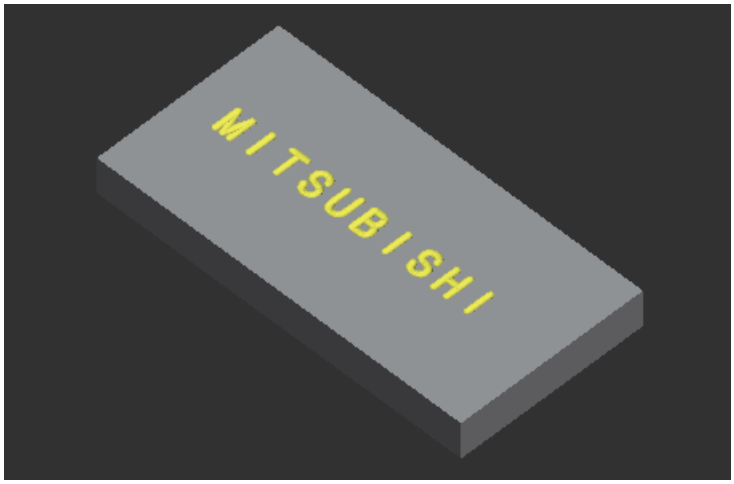
This function enables to engrave arbitrary text on the machining surface. The specified text can be engraved on the plane or cylindrical surface. The size and angle of the text can be specified using specific addresses. The text thickness is determined by the tool tip shape and the engraving depth.

(Machining example)
- The lower left point of the workpiece is the zero point in the workpiece coordinate system.
- Machining center system
- Workpiece size: W200*D100*H20 (mm)
- Tool radius: 2 (mm)
- Retraction height R: 2.5 (mm)
- Engraving depth D: 1 (mm)
- Text height H: 20 (mm)
- Text interval L: 5 (mm)
- Text to be machined: [MITSUBISHI]

Program example

G91 G28 Z0 ;	Zero point return of Z axis
G91 G28 X0 Y0 ;	Zero point return of X and Y axes
G90 G0 G54 G17 X0. Y0. ;	Workpiece coordinates and X-Y plane designation
G0 G43 H1 Z50. F3000 ;	Tool length compensation and approach to workpiece
G135 X20. Y50. Z0. R2.5 D1. H20. L5. [MITSUBISHI] ;	Text engraving cycle
G91 G0 Z50. ;	Tool retraction
G91 G28 Z0 ;	Zero point return of Z axis
G91 G28 X0 Y0 ;	
M30 ;	End command

Machining result



12.1.4 Mirror Image

12.1.4.1 Mirror Image by Parameter Setting

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

A parameter is used to designate the axis for which the mirror image function is to be executed before the machining program is run.
When mirror image is set to ON by the parameter, an operation which is symmetrical on the left and right or at the top or bottom is performed.
Each axis has its own parameter.

12.1.4.2 Mirror Image by External Input

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

Signals from an external device (PLC) to request the mirror image operation either while a machining program is running or before it is run.
When ON has been set for mirror image from an external device, an operation which is symmetrical on the left and right or at the top or bottom is performed.
Each axis has its own request signal.

12.1.4.3 Mirror Image by G Code

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	—	—	—	—	—	—	—

Using a program for the left or right side of a shape, this function can machine the other side of the shape when a left/right symmetrical shape is to be cut.
Mirror image can be applied directly by a G code when preparing a machining program.

G code	Function
G50.1	G code mirror image cancel
G51.1	G code mirror image ON

The program format for the G code mirror image is shown below.

```
G51.1  Xx1 Yy1 Zz1  ;
G51.1                 : Mirror image on
Xx1,Yy1,Zz1           : Command axes and command positions
```

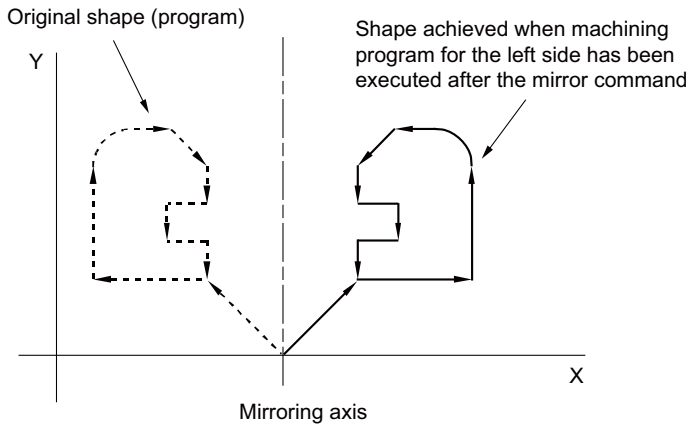
With the local coordinate system, the mirror image is applied with the mirror positioned respectively at x1, y1 and z1.

The program format for the G code mirror image cancel is shown below.

```
G50.1  Xx1 Yy1 Zz1  ;
G50.1                 : Mirror image cancel
Xx1,Yy1,Zz1           : Command axes
```

The x1, y1 and z1 indicate the axes for which the mirror image function is to be canceled and the coordinate position is ignored.

[In the case of G51.1 Xx1 ;]



12.1.4.4 Mirror Image for Facing Tool Posts

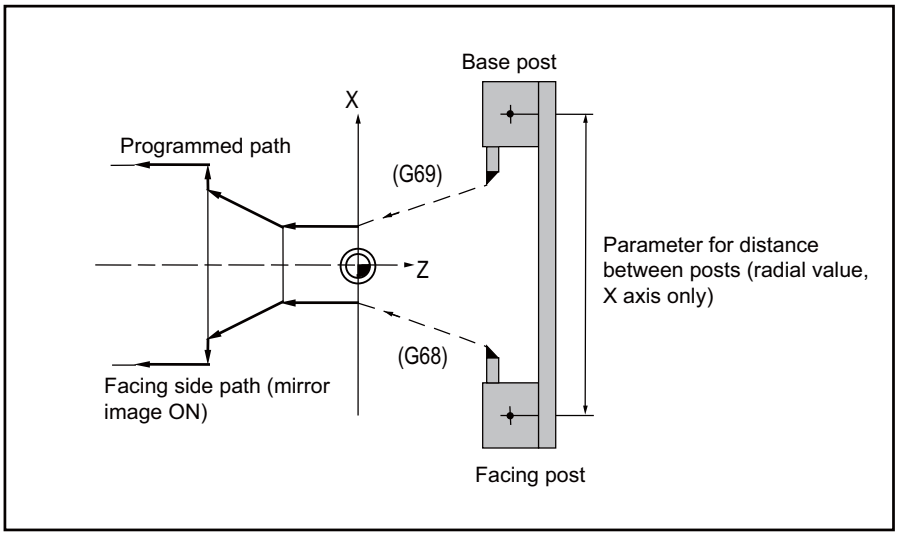
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	O	O	O	O	O	O	O

With machines in which the base tool post and facing tool post are integrated, this function enables the programs prepared for cutting at the base side to be executed by the tools on the facing side.
The distance between the two posts is set beforehand with the parameter.
The command format is as follows.

G code	Function
G68	Facing tool post mirror image ON
G69	Facing tool post mirror image OFF

When the G68 command is issued, the subsequent program coordinate systems are shifted to the facing side and the movement direction of the X axis is made the opposite of that commanded by the program. When the G69 command is issued, the subsequent program coordinate systems are returned to the base side.

The facing tool post mirror image function can be set to ON or OFF automatically by means of T (tool) commands without assigning the G68 command.
A parameter is used to set ON or OFF for the facing tool post mirror image function corresponding to the T commands.



12.1.4.5 T Code Mirror Image for Facing Tool Posts

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	O	O	O	O	O	O	O

If, when tools that correspond to tool Nos. 1 to 64 are selected (T commands), these are tool Nos. for which the facing tool post mirror image function has already been designated with a parameter, the status equivalent to G68 (facing tool post mirror image ON) is established. When the commands apply to tool Nos. for which the facing tool post mirror image function is not designated, the status equivalent to G69 (facing tool post mirror image OFF) is established.

12.1.5 Coordinate System Operation

12.1.5.1 Coordinate Rotation by Program

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	—

Coordinate rotation by program is used to rotate the programmed shape by the specified angle and to machine it. Even complex shapes can be easily machined in the rotated position.

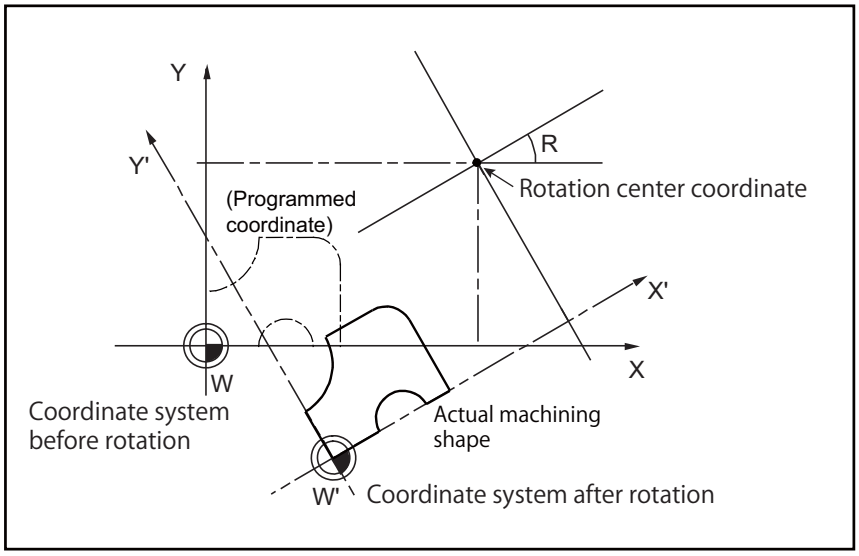
[M system]

G68 X__ Y__ R__ ;	**Coordinate rotation ON**

G69 ;	**Coordinate rotation cancel**

X,Y	Rotation center coordinates
R	Rotation angle

Example



[L system]

G68.1 X__ Z__ R__ ;	**Coordinate rotation ON**

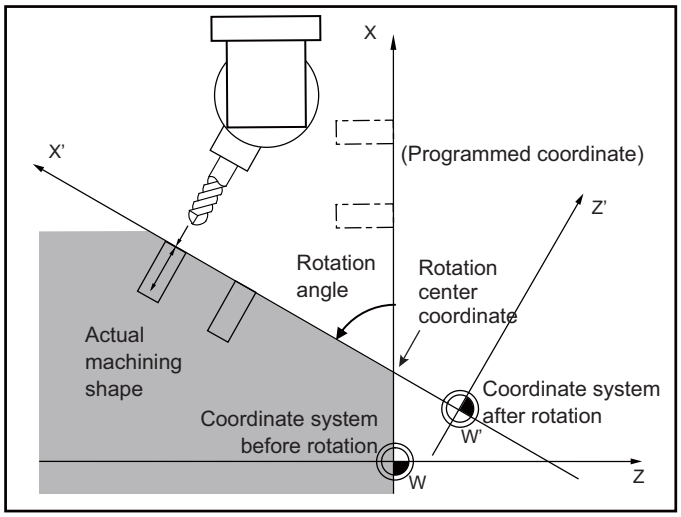
G69.1 ; Coordinate rotation cancel	**Coordinate rotation cancel**

X,Y	Rotation center coordinates
R	Rotation angle

Note

(1) This function rotates the coordinate system and realizes mainly drilling and tapping.

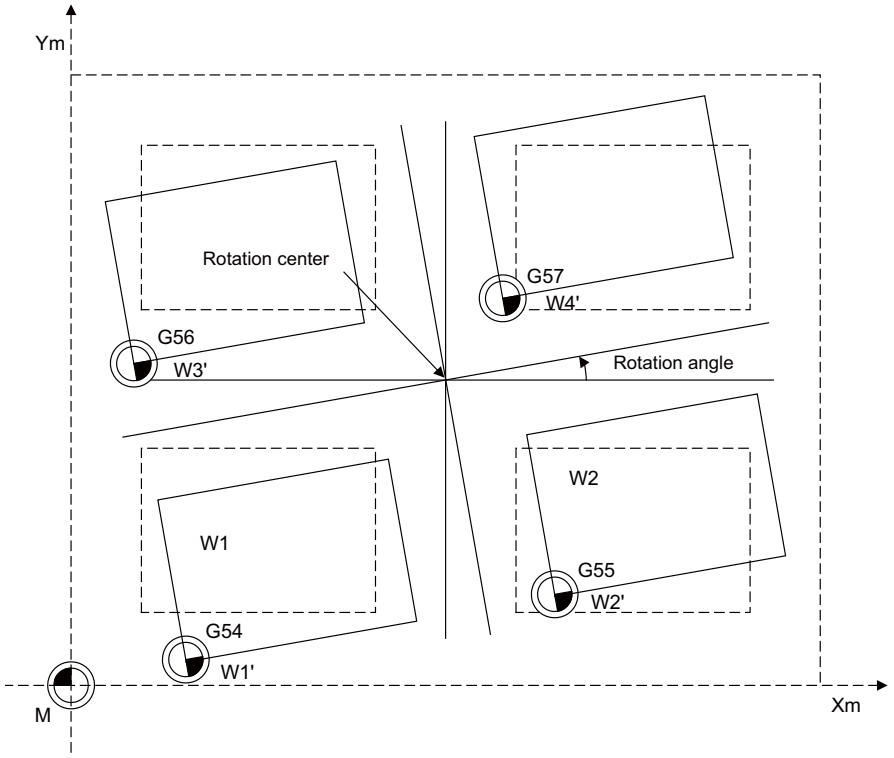
Example



12.1.5.2 Coordinate Rotation by Parameter

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	—	—	—
L	—	—	—	—	—	—	—

If a deviation occurs between the workpiece alignment line and machine coordinate system's coordinate axis when the workpiece is mounted, the machine can be controlled by rotating the machining program coordinates according to the workpiece alignment line deviation. The coordinate rotation amount and the center of the rotation can be set with the parameters. The coordinate rotation amount can also be set with the G10 command.

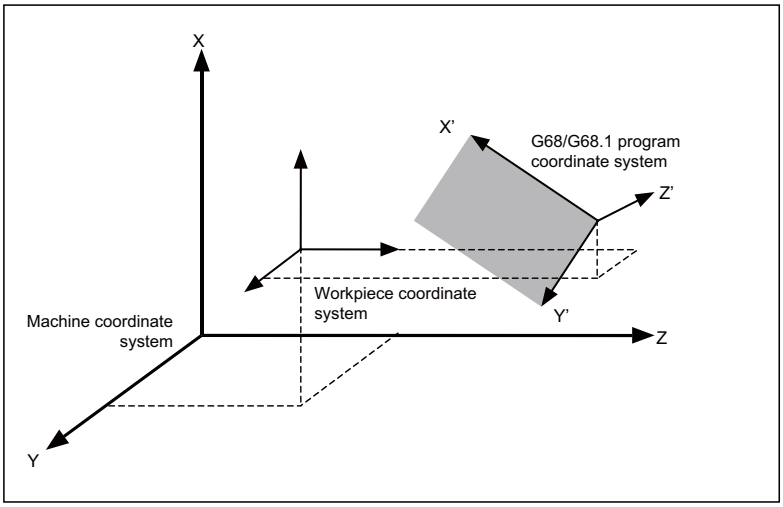


- As for the rotation center coordinate position, designate the position on the machine coordinate system.
- All workpiece coordinate systems from G54 to G59, G54.1 rotate with the rotation command.
  The machine coordinate system does not rotate.
- If the setting is made on the parameter screen, the setting value will be valid at the timing of the followings:
  Automatic operation: The setting values become valid from the next block after parameter settings have been configured.
  Manual operation: After the parameter is set, the setting value will be valid if the PLC signal (Coordinate rotation by parameter: Manual feed coordinate system) is ON.

12.1.5.3 3-Dimensional Coordinate Conversion

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	—
L	Δ	Δ	Δ	Δ	○	○	—

With the 3-dimensional coordinate conversion function, a new coordinate system can be defined by rotating and moving in parallel the zero point in respect to the X, Y and Z axes of the currently set workpiece coordinate system. By using this function, an arbitrary spatial plane can be defined, and machining on that plane can be carried out with normal program commands.



When the G68 (M system)/G68.1 (L system) command is issued, the zero point is shifted by the command value (x, y, z) in respect to the current local coordinate system. A new G68/G68.1 program coordinate system rotated by the designated rotation angle r in respect to the commanded rotation center direction (i, j, k) is created.
The local coordinate system is the same as the workpiece coordinate system when the local coordinate system offset is not ON.

The program format is as follows.

[M system]

G68 X__ Y__ Z__ I__ J__ K__ R__ ;	
G68	: 3-dimensional coordinate conversion mode command
X, Y, Z	: Rotation center coordinates
I,J,K	: Rotation center axis direction
R	: Rotation angle

G69 ;	
G69	: 3-dimensional coordinate conversion mode cancel command

[L system]

G68.1 X__ Y__ Z__ I__ J__ K__ R__ ,E__ ;	
G68.1	: 3-dimensional coordinate conversion mode command
X, Y, Z	: Rotation center coordinates
I,J,K	: Rotation center axis direction
R	: Rotation angle
,E	: Command coordinate system selection

G69.1 ;	
G69.1	: 3-dimensional coordinate conversion mode cancel command

12.1.6 Dimension Input

12.1.6.1 Corner Chamfering/Corner R

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

This function executes corner processing by automatically inserting a straight line or an arc by the commanded amount between two consecutive movement blocks (G01/G02/G03).
The corner command is executed by assigning the ",C" or ",R" command for the block at whose end point the corner is inserted.

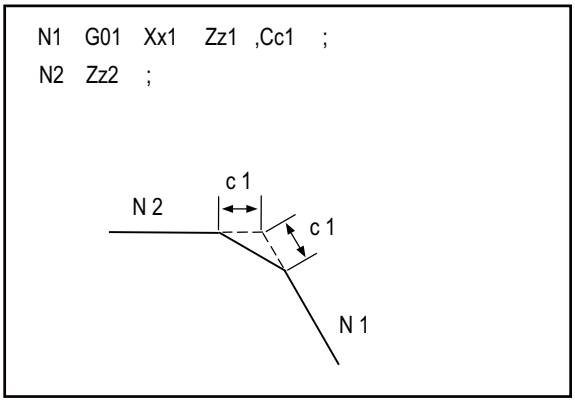
Corner chamfering I/ Corner R I

When ",C" or ",R" is commanded for linear interpolation, corner chamfering or corner R can be inserted between linear blocks.

Corner chamfering I

Example

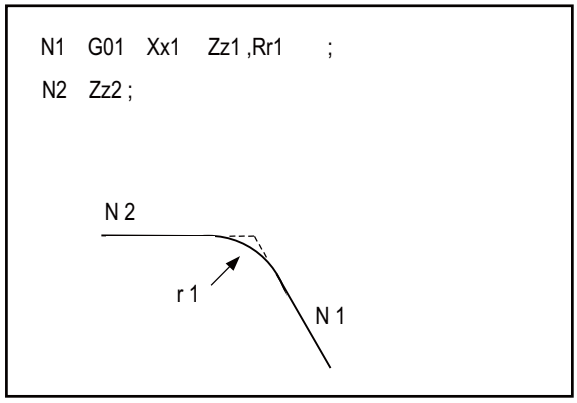
N1 G01 Xx1 Zz1 ,Cc1 ;
N2 Zz2 ;



Corner R I

Example

N1 G01 Xx1 Zz1 ,Rr1 ;
N2 Zz2 ;



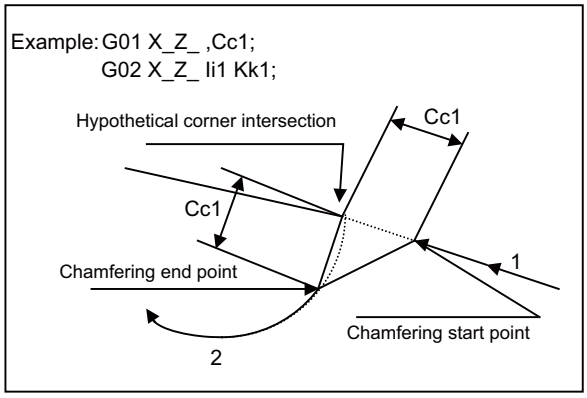
Note

(1) If a corner chamfering or corner R command is issued specifying a length longer than the N1 or N2 block, a program error occurs.

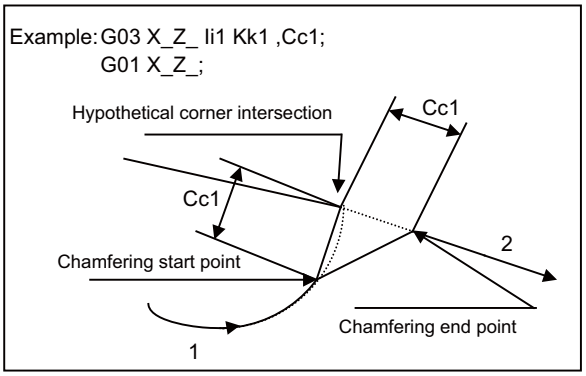
Corner chamfering II/ corner R II

When ",C" or ",R" is commanded in a program between linear-circular, corner chamfering or corner R can be inserted between blocks.
When the parameters are set, "I_", "K_", "C_" can be used to command chamfering instead of ",C", and "R_" can be used to command rounding instead of ",R_". (L system only)

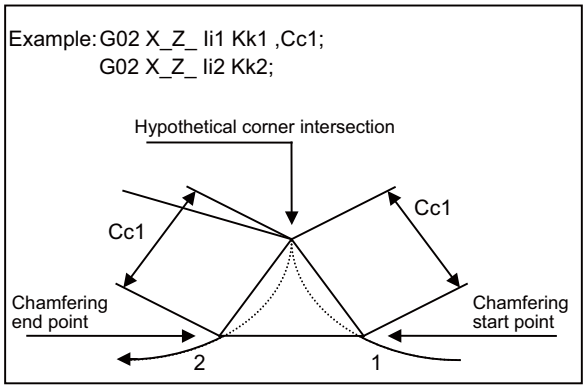
■ Corner chamfering II (Linear - circular)



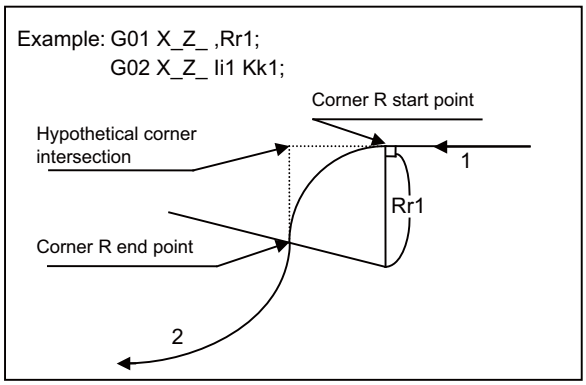
■ Corner chamfering II (Circular - linear)



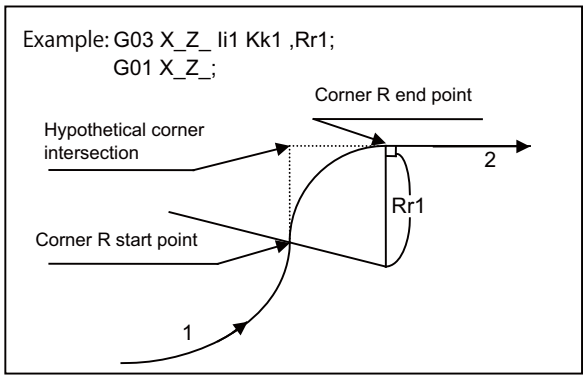
■ Corner chamfering II (Circular - circular)



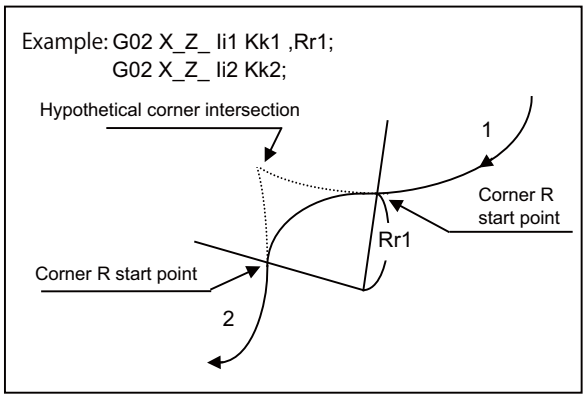
■ Corner R II (Linear - circular)



■ Corner R II (Circular - linear)



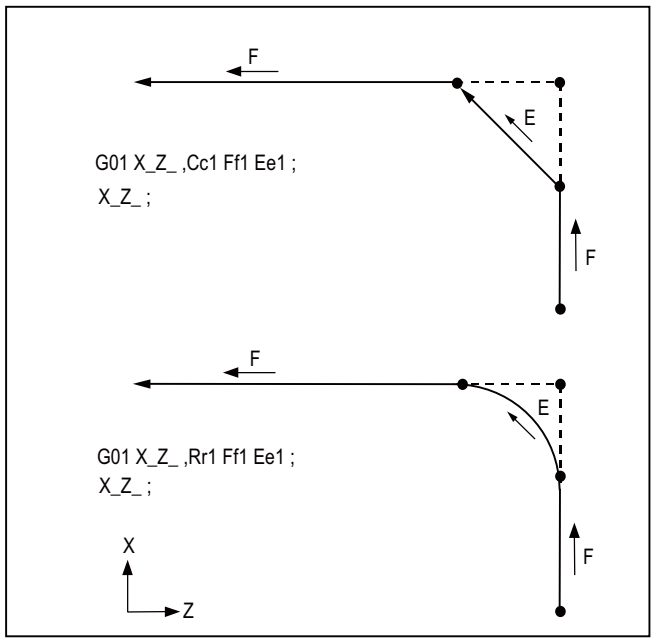
■ Corner R II (Circular - circular)



Specification of corner chamfering expansion/corner R expansion

An E address can be used to specify the speed for corner chamfering or corner R. This enables a corner to be cut to a correct shape.

(Example)



An E address is a modal It remains effective for feeding in next corner chamfering or corner R.
An E address has two separate modals: synchronous and asynchronous feed rate modals. The effective feed rate is determined by synchronous (G95) or asynchronous (G94) mode.
If an E address is specified in 0 or no E command has been specified, the feed rate specified by an F command is assumed as the feed rate for corner chamfering or corner R.
An E address modal is not cleared even if the reset button is pressed.
It is cleared when the power is turned OFF. (In the same manner as F commands.)

Corner chamfering/corner R (I, K designation) (L system only)

With this command format, by means of parameter settings, corners are chamfered using the "I", "K" or "C" address without a comma, and corners are rounded using the "R" address.
The ",C" and ",R" addresses with commas can also be used.

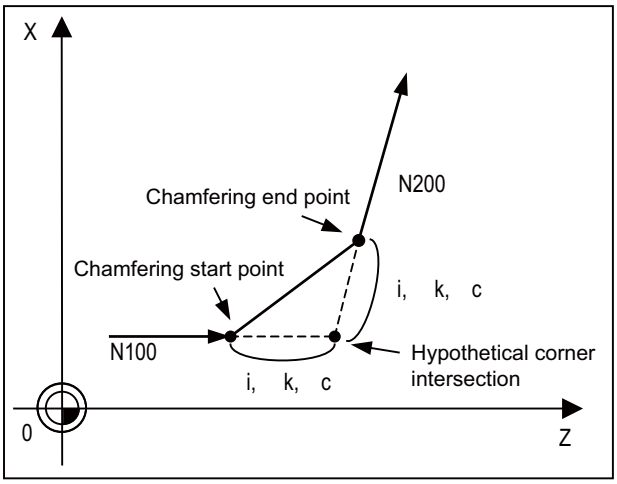
■ Corner chamfering (I, K designation)

Corners are chamfered using the "I_", "K_" or "C_" address without a comma, ",".
Corners can be chamfered to any angle.
Signs, if they are provided for the corner chamfering commands, are ignored.

[Command format]

```
N100  Xx1/Uu1 Zz1/Ww1 Ii1/Kk1/Cc1 ;
N200  Xx1/Uu1 Zz1/Ww1 ;
```

Xx1/Uu1	X axis end point coordinate
Zz1/Ww1	Z axis end point coordinate
Ii1/Kk1/Cc1	The length from the hypothetical corner intersection to the chamfering start point or chamfering end point is designated using the "I", "K" or "C" address.



- If multiple "I", "K" or "C" addresses or duplicated addresses have been designated in the same block, the last address will take effect.
- If both corner chamfering and corner R commands are present in the same block, the last address will take effect.
- If "C" is used as the name of an axis, corner chamfering commands cannot be designated using the "C" address.
- If "C" is used as a 2nd miscellaneous function, corner chamfering commands cannot be designated using the "C" address.
- Corner chamfering commands using the "I" or "K" address cannot be designated in an arc command block. "I" and "K" are the arc center commands.

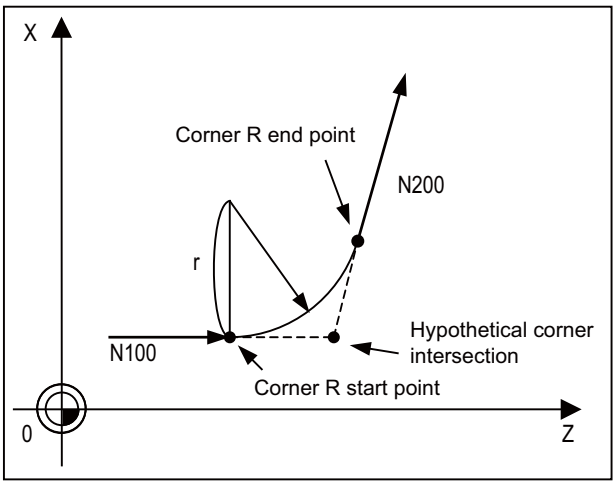
■ Corner R ("I", "K" designation)

Corners are rounded using the "R_" address without a comma, ",".
Corners can be rounded to any angle.
Signs, if they are provided for the corner R commands, are ignored.

[Command format]

```
N100 Xx1/Uu1 Zz1/Ww1 Rr1 ;
N200 Xx1/Uu1 Zz1/Ww1 ;
```

Xx1/Uu1	X axis end point coordinate
Zz1/Ww1	Z axis end point coordinate
Rr1	Radius of corner R arc



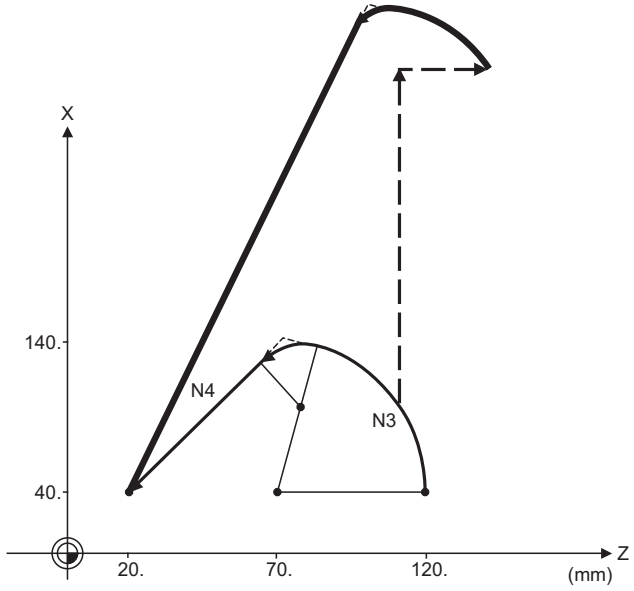
- If both corner chamfering and corner R commands are present in the same block, the last address will take effect.
- Corner R commands using the "R" address cannot be designated in an arc command block. "R" is regarded as the arc radius command in such a block.

Interrupt during corner chamfering/Interrupt during corner R

Shown below are the operations of manual interruption during corner chamfering or corner R.

■ With an absolute command and manual absolute switch ON:

```
N1 G28 XZ;
N2 G00 X40. Z120.;
N3 G03 X140.Z70. K-50. ,R20. F100 ;
N4 G01 X40. Z20.
      :
```



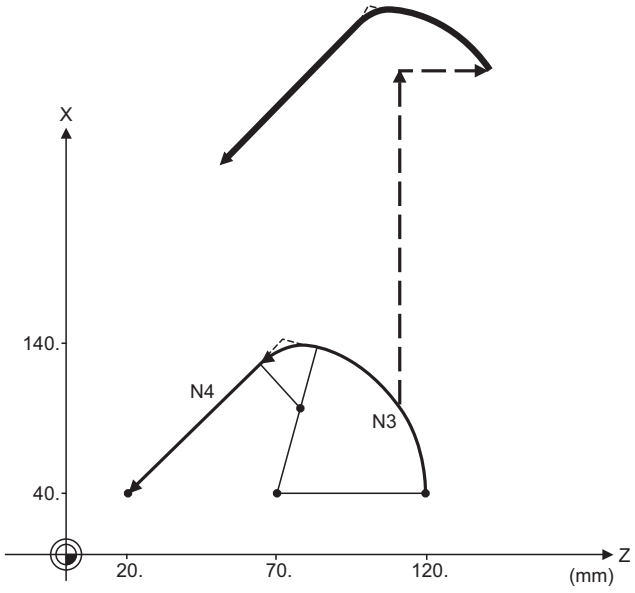
Amount of interruption

Path with interruption

Path without interruption

■ With an incremental command or manual absolute switch OFF:

```
N1 G28 XZ;
N2 G00 U40. W120.;
N3 G03 U100. W-50. K-50. ,R20. F100 ;
N4 G01 U-100.W-50. ;
      :
```



Amount of interruption

Path with interruption

Path without interruption

12.1.6.2 Linear Angle Command

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

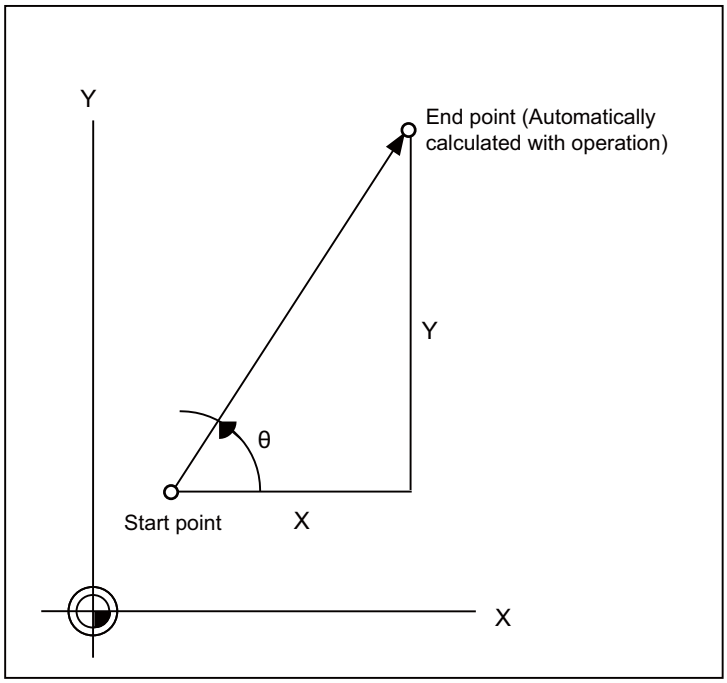
The end point coordinates are automatically calculated by assigning one element (one component of the selected plane) of the end point coordinates and the linear angle.

```
G17 Xx1 Aa1 ;
```

```
G17 Yy1 Aa1 ;
```

G17	Plane selection
Xx1,Yy1	1 element of the end point coordinate
Aa1	Angle

Example



Note

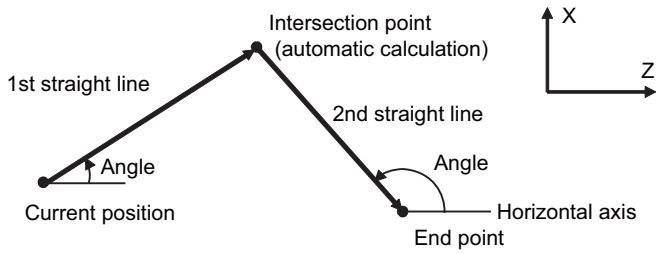
(1) If the axis "A" or 2nd miscellaneous function "A" is used, address "A" is treated as the axis "A" command or the 2nd miscellaneous function, respectively.

12.1.6.3 Geometric Command

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	○	○	○	○	○	○	○

■ Geometric I

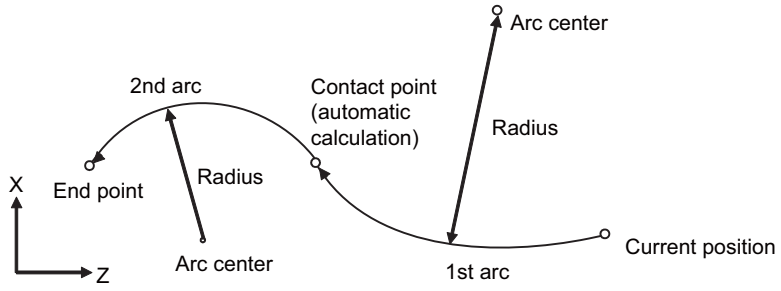
When it is difficult to calculate the intersection point of two straight lines in a continuous linear interpolation command, the end point of the first straight line will be automatically calculated inside the CNC and the movement command will be controlled, provided that the slope of the first straight line as well as the end point coordinates and slope of the second straight line are commanded.



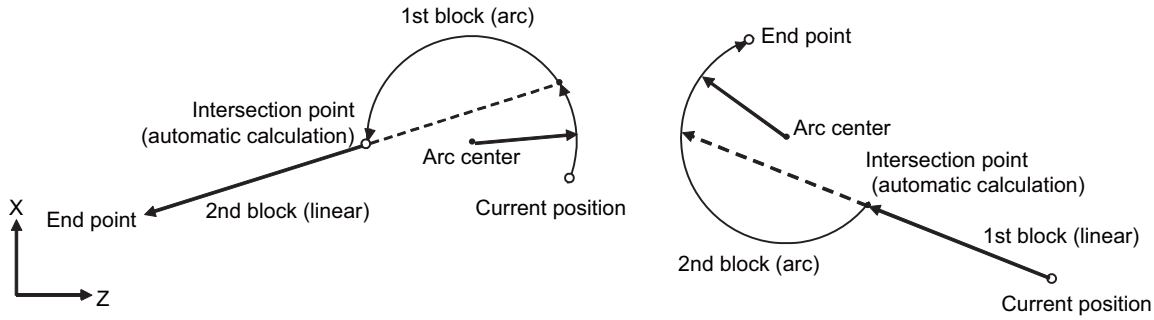
■ Geometric IB

With the geometric IB function, the contact and intersection are calculated by commanding an arc center point or linear angle in the movement commands of two continuous blocks (only blocks with arc commands), instead of commanding the first block end point.

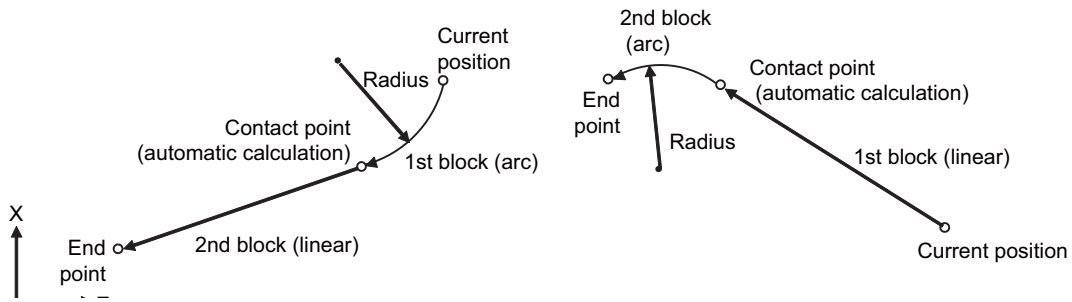
[Two-arc contact]



[Linear-arc (arc-linear) intersection]



[Linear-arc (arc-linear) contact]



12.1.6.4 Polar Coordinate Command

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	—	—	—	—	—	—	—

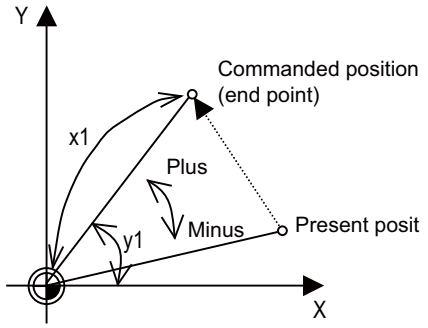
With this function, the end point position is commanded with the radius and angle.

■ Command format

G16 ;	**Polar coordinate command mode ON**

G15 ;	**Polar coordinate command mode OFF**

■ Program example

G1x ; G16 ;	Plane selection for polar coordinate command (G17/G18/G19) Polar coordinate command mode ON
G9x G01 Xx1 Yy1 F2000 ; :	Polar coordinate command G9x: Center selection for polar coordinate command (G90/G91)) G90: The workpiece coordinate system zero point is the polar coordinate center. G91: The current position is the polar coordinate center. x1: 1st axis for the plane…The radius commanded y1: 2nd axis for the plane…The angle commanded
	 For G90/G17(X-Y plane)
G15 ;	Polar coordinate command mode OFF

12.1.7 Axis Control

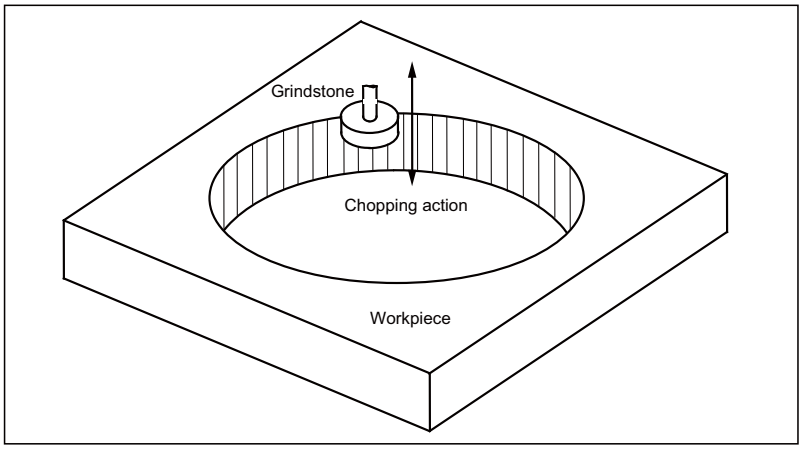
12.1.7.1 Chopping

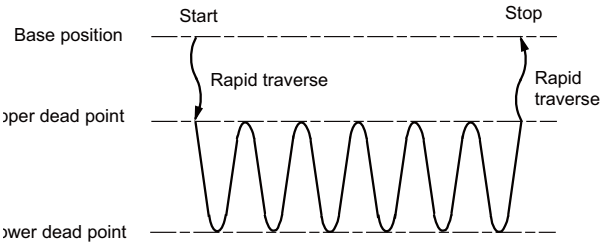
12.1.7.1.1 Chopping

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

This function continuously raises and lowers the chopping axis independently of the program operation when workpiece contours are to be cut. It can be used for grinding operations using machining centers, for instance.
Which of the axes is to serve as the chopping axis is set by a parameter beforehand.

■ Chopping operation





The chopping operation is initiated by setting the upper dead point position, lower dead point position and number of cycles (number of up/down movements per minute) and pressing the chopping start switch.

Note

(1) The upper dead point position, lower dead point position and number of cycles are set and the start and stop commands are designated by input signals from the user PLC.
(2) The setting for the number of cycles differs according to the motor, inertia and other factors.

The chopping operation is performed as follows.

- The axis moves from the base position to the upper dead point by rapid traverse.
- Next, the axis moves repeatedly from the upper dead point to the lower dead point and then from the lower dead point to the upper dead point. (Sinusoidal waveforms)
  The feed rate is tailored to achieve the number of cycles set for the up/down motion.

[Chopping override]

Override in 1% increments from 0% to 100% can be applied to the chopping operation.

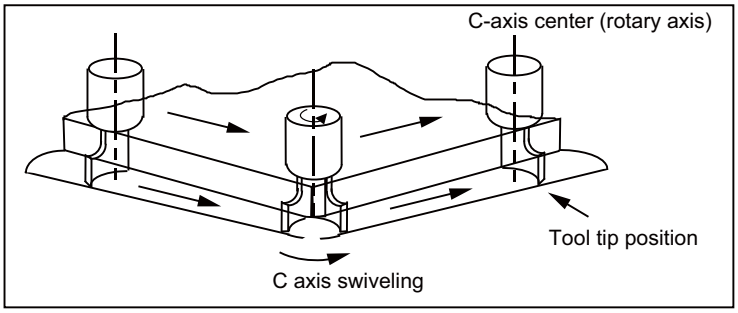
Note

(1) Bear in mind that the override increment differs according to the machine specifications.

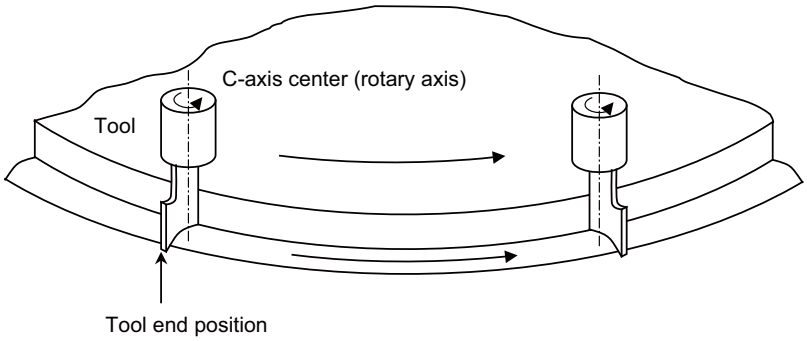
12.1.7.2 Normal Line Control

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	—
L	—	—	—	—	—	—	—

This function controls the swiveling of the C axis (rotary axis) so that the tool is always pointing in the normal line direction for the X and Y axes movement commands during program operation. It can be used for machining hale, for sewing operations involving sewing machines, carpets, etc. It is valid for only one C axis (rotary axis).
At the block seams, the C axis turning is controlled so that the tool faces the normal line direction at the next block's start point.



During arc interpolation, the C axis turning is controlled in synchronization with the operation of arc interpolation.



G code	Function
G40.1	Normal line direction control cancel
G41.1	Normal line direction control left ON
G42.1	Normal line direction control right ON

12.1.7.3 Circular Cutting

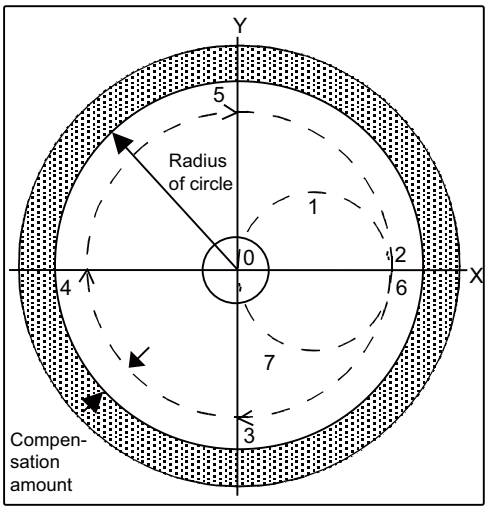
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	—	—	—	—	—	—	—

In circular cutting, a system of cutting steps are performed: first, the tool departs from the center of the circle, and by cutting along the inside circumference of the circle, it draws a complete circle, then it returns to the center of the circle. The position at which G12 or G13 has been programmed serves as the center of the circle.

■ Command format

G12/(G13) Ii Dd Ff ;	
G12	: Circular cutting command CW (clockwise)
G13	: Circular cutting command CCW (counter clockwise)
Ii	: Radius of complete circle
Dd	: Compensation No.
Ff	: Feed rate

When the G12 command is used (path of tool center)
0 -> 1 -> 2 -> 3 -> 4 -> 5 -> 6 -> 7 -> 0
When the G13 command is used (path of tool center)
0 -> 7 -> 6 -> 5 -> 4 -> 3 -> 2 -> 1 -> 0



Note

(1) Circular cutting is carried out on the plane which has been currently selected (G17, G18 or G19).
(2) The (+) and (-) signs for the compensation amount denote reduction and expansion respectively.

12.1.7.4 Vibration Cutting Control

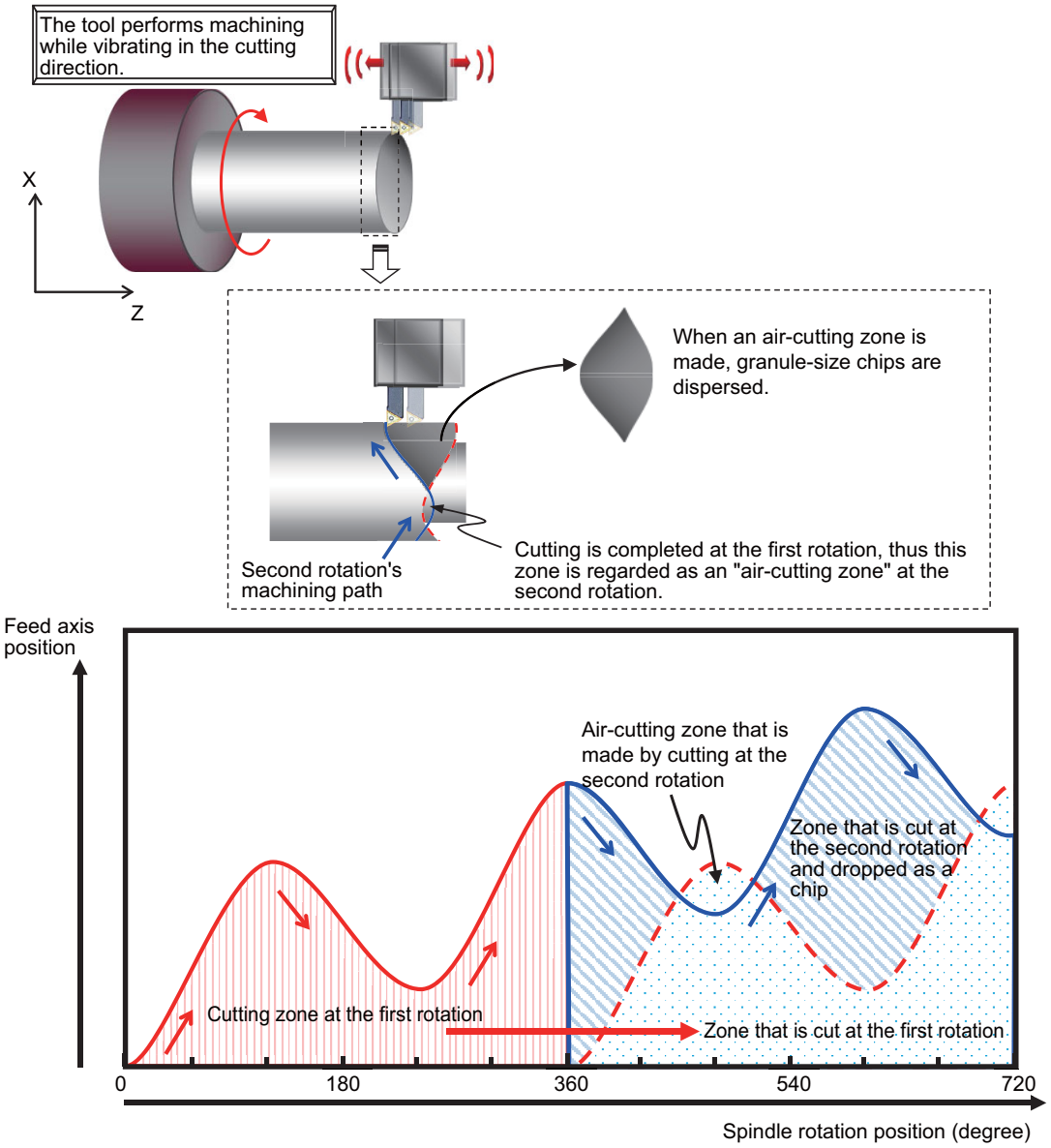
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	□	□	□	□	□
L	□	□	□	□	□	□	□

While cutting machining is performed, the chips can be dispersed by applying vibration in synchronization with the spindle rotation to the feed axis so that an air-cutting zone is made. Difficult-to-cut workpieces such as stainless steel can be machined easily and at high speed. In addition, this function can reduce the heat generated by machining and extend tool life.

Note

(1) The vibration cutting extension unit is required.
(2) The vibration axis is the only one in the commanded part system. (Only the specified single axis vibrates even for the cutting command of two or more axes such as a taper.)
(3) This function may affect the surface quality of the workpiece as the axis vibration. Use it for the machining such as rough machining.

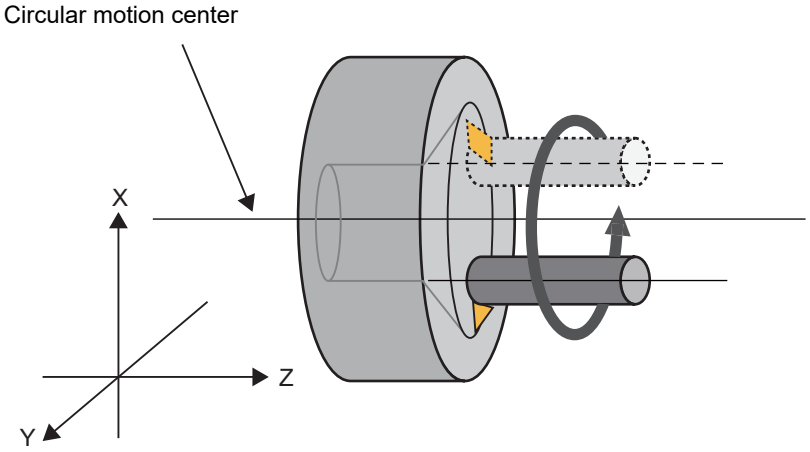
[Lathe system]



12.1.7.5 Orbit Machining

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	△	△	△	△	○	○	—
L	—	—	—	—	—	—	—

This function allows you to perform machining equivalent to turning on a lathe using a machining center.
The circular motion is performed while the tool nose is vertically pressed against the machining surface by synchronizing the three basic axes with the rotation of the turning tool.
This function enables finish machining of different inner or outside diameter of a workpiece without changing the turning tools.



■ Command format

Orbit machining (G08.6) is a non-modal command of the G group 0.
The orbit machining command must be issued alone in a block. If another command is in the same block, the program error occurs.

Orbit machining mode start

G08.6 Q__ R__;

Q	Q1: Internal machining Q2: Outside machining
R	Tool nose angle

Orbit machining mode cancel

G08.6 Q0;

Q	Q0: Machining mode cancel

12.1.8 Multi-Part System Control

12.1.8.1 Timing Synchronization Between Part Systems

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

Multiple machining programs can be operated independently at same time for multi-axis and multi-part system mixed control CNC. This function enables the timing synchronization between part systems or the operation of only one part system while the machining programs are being operated independently for each part system.

■ Command format

Timing synchronization with ! code

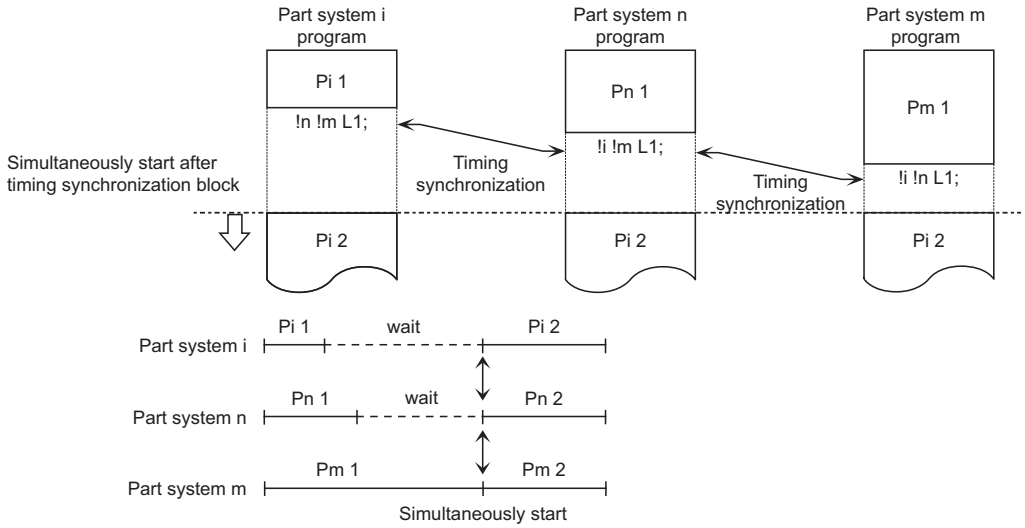
!n (!m...)L __ ;
!n !m : Part system number for timing synchronization (Multiple command can be performed.)
Follows the parameter if the part system number is omitted.
L : Timing synchronization number (0 to 9999)

Timing synchronization with ! code is also valid among three part systems and more.

Timing synchronization with M code

M* ;**
*** : M code for timing synchronization

The range of M code for the timing synchronization is determined with the minimum and maximum values set with the parameter beforehand.



Note

(1) The timing synchronization can be disvalued with the parameter setting and "Timing synchronization between part systems ignored" signal. When a part system to be waited for a simultaneous operation is not operating, the currently operating part system will move on to the next block without executing timing synchronization. This function is useful when conducting a program check on each part system.
(2) The timing synchronization command is normally assigned as a separate command. However, if a movement command and M, S or T command have been assigned in the same block, a parameter is set to decide whether timing synchronization is executed upon executing of the movement command and M, S or T command, or the movement command and M, S or T command are executed upon completion of timing synchronization.

12.1.8.2 Start Point Designation Timing Synchronization

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	—
L	○	○	○	○	○	○	○

Multiple machining programs can be operated independently at same time for multi-axis and multi-part system mixed control CNC. This function enables the timing synchronization with the position in the movement block while the machining programs are being operated independently for each part system.
When a workpiece is being delivered from part system 1 to part system 2, the movement operation in part system 2 can be started in the middle of the part system 1 tool post movement with this function so that the machining time can be reduced.

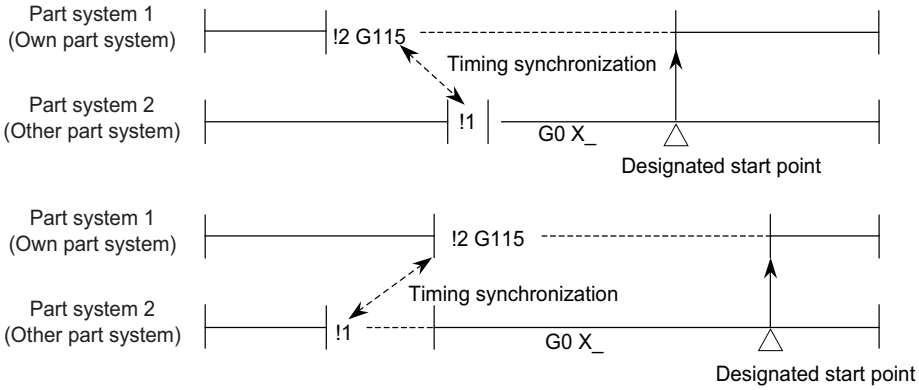
A part system for start point designation timing synchronization can be designated by adding ! code on the command block G115 and G116.

Start point designation timing synchronization Type 1 (G115)

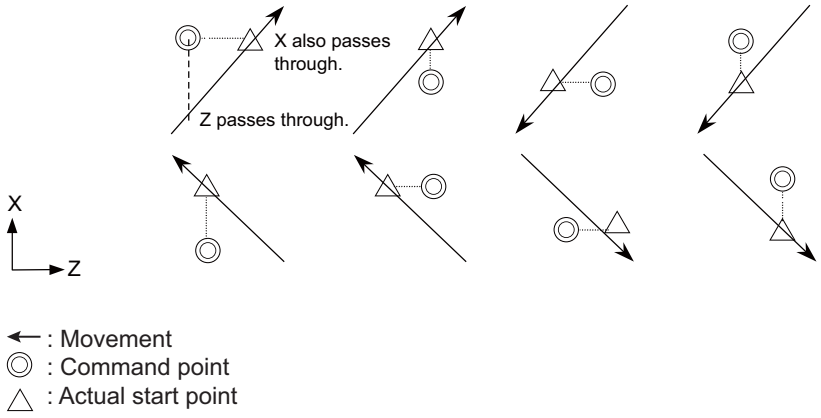
■ Command format

!n L __ G115 X __ Z __ C __ ;		
!n	:	Part system number for timing synchronization
L	:	Timing synchronization number (0 to 9999)
G115	:	G command
X Z C	:	Start point (Designated with the axis and workpiece coordinate value of other part system.)

- The other part system starts first when timing synchronization is executed.
- The own part system waits for the other part system to move and reach the designated start point, and then starts.



- When the start point designated by G115 is not on the next block movement path of the other part system, the own part system starts once the other part system has reached each start point axis coordinate position.

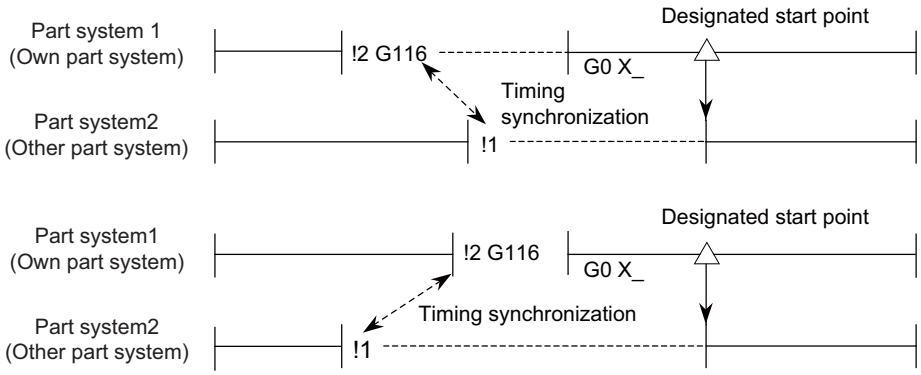


Start point designation timing synchronization Type 2 (G116)

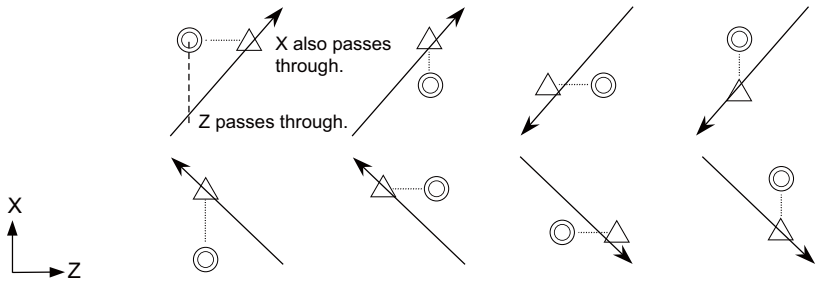
■ Command format

!n L __ G116 X __ Z __ C __ ;	
!n	: Part system number for timing synchronization
L	: Timing synchronization number (0 to 9999)
G116	: G command
X Z C	: Start point (Designated the axis and workpiece coordinate value of the own part system.)

- The own part system starts first when timing synchronization is executed.
- The other part system waits for the own part system to move and reach the designated start point, and then starts.



- When the start point designated by G116 is not on the next block movement path of the own part system, the other part system starts once the own part system has reached each start point coordinate position.



←: Movement
◎ : Command point
△ : Actual start point

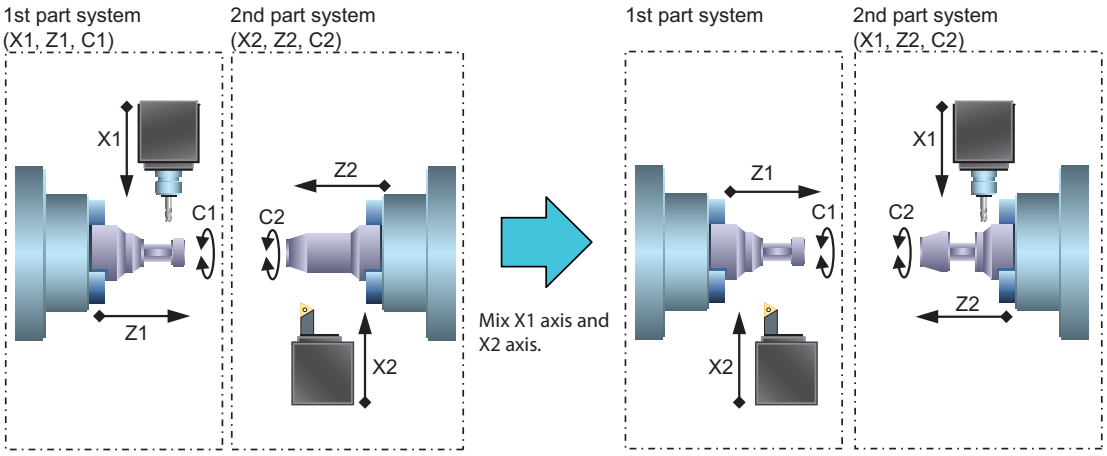
12.1.8.3 Mixed Control

12.1.8.3.1 Mixed Control (Cross Axis Control)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	Δ	Δ	Δ	Δ	○	○	—

Performs the mixed control (cross axis control) by G command or PLC signal.

Any axis to be replaced by another axis between part systems.
This makes it possible to perform operations which are not possible with regular axis configurations; for instance, tools which are provided only on part system 1 can be used for machining on part system 2.
This function cannot be used with arbitrary axis exchange control.

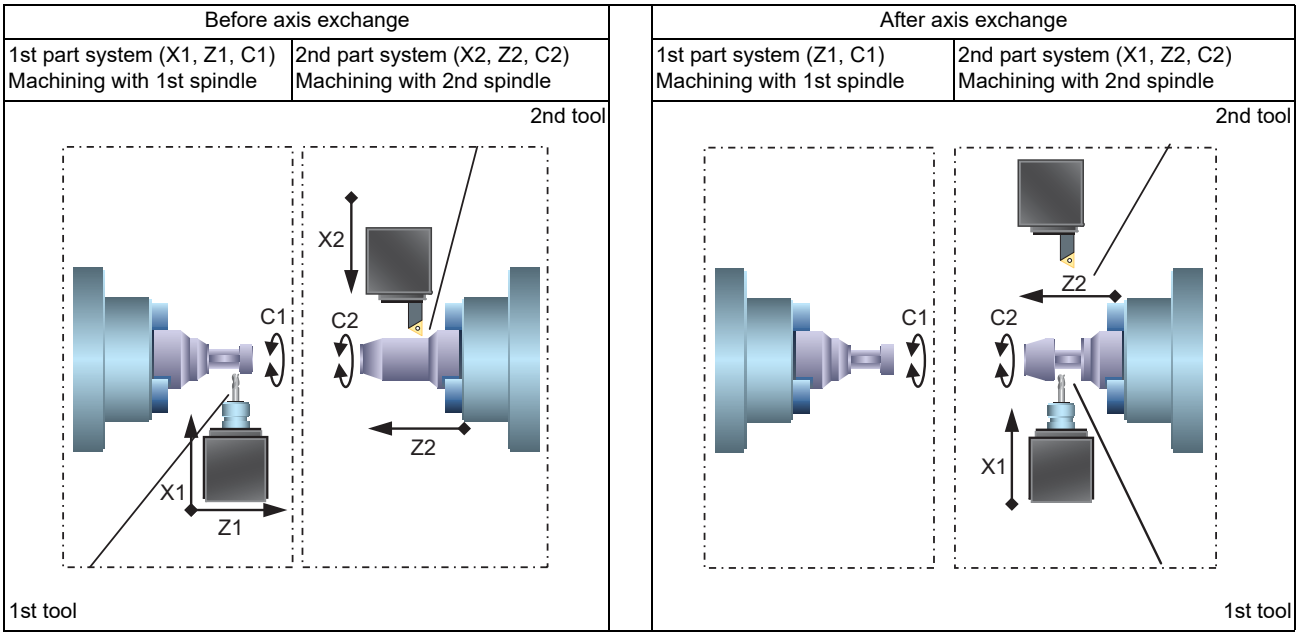


12.1.8.3.2 Arbitrary Axis Exchange Control

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	—

An arbitrary axis can be exchanged freely between part systems in the multiple part systems.
The machining can be freer in the multiple part systems by exchanging an axis which can be commanded for machining programs in each part system.
This makes it possible to perform operations which are not possible with regular axis configurations; for instance, tools which are provided only on the 1st part system can be used for machining on the 2nd part system.
This function cannot be used with the mixed control (cross control).

[Using X1 axis in part system 2]



■ Command format

Arbitrary axis exchange command
Commands the arbitrary axis exchange by G140.

G140 command address = axis address command address = axis address...;	
Command address	: Designate the command address used in commands such as the movement command with one alphabetical character set to the parameter after the arbitrary axis exchange command (G140).
Axis address	: Designate the axis name for arbitrary axis exchange with two alphanumeric characters set to the parameter.

Arbitrary axis exchange return command
Returns the control right of the axis, exchanged by the previous arbitrary axis exchange command (G140) in the commanded part system, to the state before the axis exchange.

```
G141 ;
```

Reference axis arrange return command
Returns the control right of the axis, exchanged by the arbitrary axis exchange command (G140) in the commanded part system, to the power-on state.

```
G142 ;
```

Note

(1) For G code list 6 and 7, replace G140, G141 and G142 with G165, G166 and G167.

12.1.8.4 Control Axis Superimposition

12.1.8.4.1 Control Axis Superimposition

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	Δ	Δ	Δ	Δ	○	○	—

In the multi-part system, an axis in a part system can be superimposed on an axis in another part system and can be controlled. This is effective when machining by the 1st part system (X1, Z1) and the 2nd part system (X2, Z2) are executed simultaneously for the machine configuration where a workpiece moves by a movement command to the Z axis direction. The machining of the reference axis part system (the 1st part system) and the superimposed axis (the 2nd part system) can be operated simultaneously with no need to fix the workpiece position for machining in the superimposed part system (the 2nd part system).

Reference axis: The basic in the control axis superimposition function (moves only by its own axis command)

Superimposed axis: The axis which moves including the reference axis movement in the control axis superimposition function (moves by reference axis or/and own axis command).

The control axis superimposition can be commanded with G code or PLC signal.

■ Command format

Superimposition start command

G126 Superimposed axis name = Reference axis name ",P"__ ;

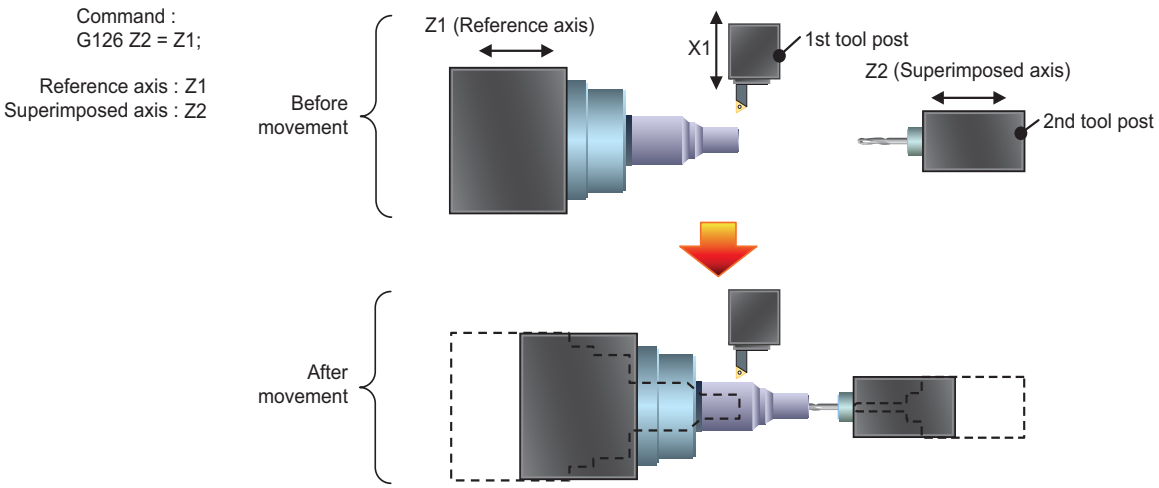
Superimposed axis name	: Specify the superimposed axis name for the superimposition control with 2 alphanumeric characters set to the parameter.
Reference axis name	: Specify the reference axis name for the superimposition control with 2 alphanumeric characters set to the parameter. Put the symbol "-" at the beginning of the reference axis name for synchronizing the superimposed axis with the reference axis in the reverse direction.
, (comma)	: Specify as delimiter when the address is needed to be assigned after "superimposed axis name = reference name".
P	: Superimposed axis workpiece coordinate system designation (The command range of the coordinate value is mm/inch. The decimal point command is valid.)

Superimposition end command

G126 Superimposed axis name;

Superimposed axis name	: Specify the name of the superimposed axis which is working for the superimposition control with 2 alphanumeric characters set to the parameter.

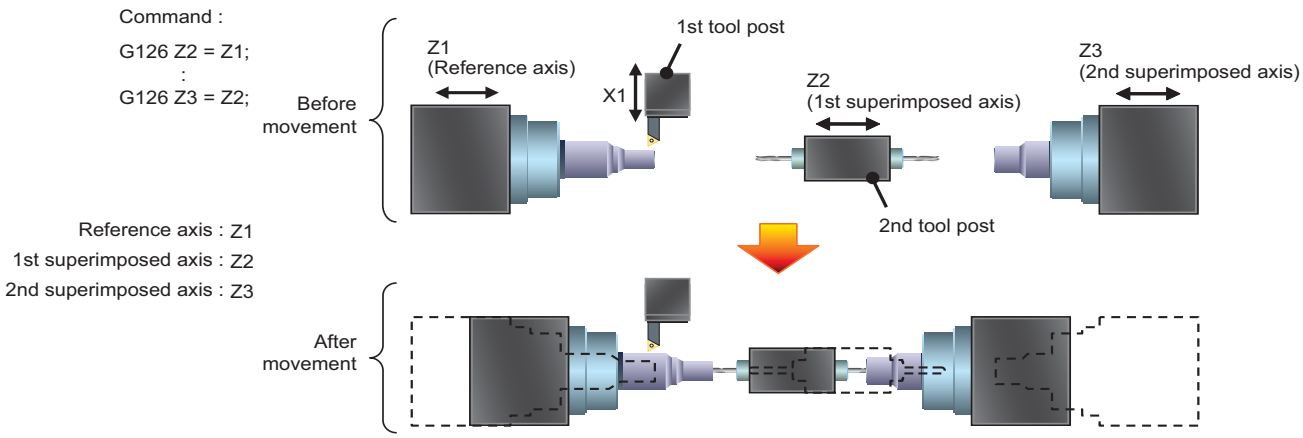
While using the control axis superimposition function, the workpiece zero point of the superimposed axis moves according to the movement amount of the reference axis. Because the superimposed axis tries to maintain the position in the workpiece coordinate system, it moves as much as the movement amount of the reference axis.



■ Superimposition control among 3 axes

Superimposition can be controlled among 3 axes.

3 axes tandem superimposition is the condition that an axis is the superimposed axis and becomes the reference axis for another superimposed axis at same time. The first superimposed axis is "1st superimposed axis" and the other axis is "2nd superimposed axis" for 3 axes tandem superimposition.



12.1.8.4.2 Arbitrary Axis Superimposition Control

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	Δ	Δ	Δ	Δ	—	—	—

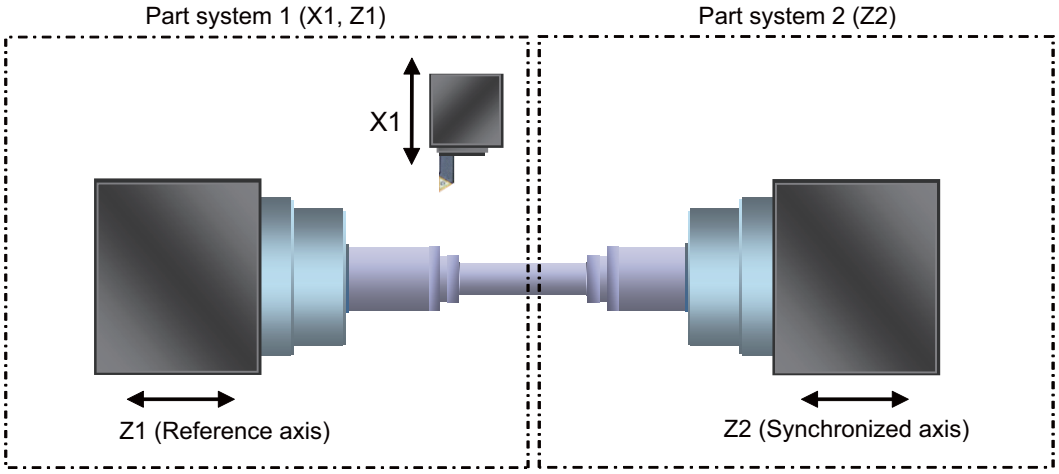
The arbitrary control axis in other part system can be moved by superimposing on the movement command for the arbitrary control axis in own part system.

12.1.8.5 Control Axis Synchronization Between Part Systems

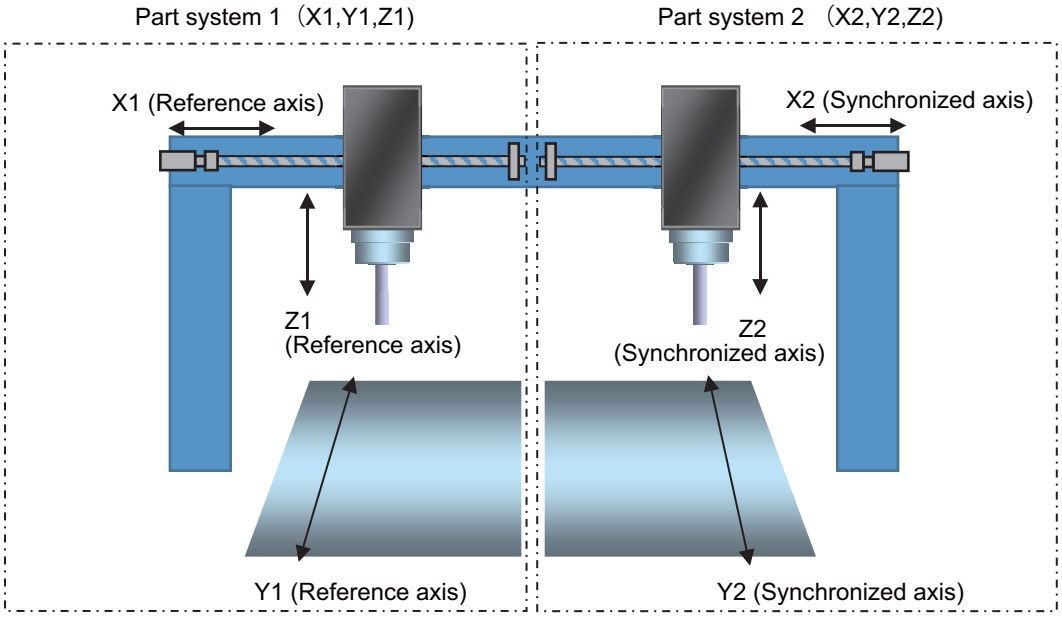
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	—
L	Δ	Δ	Δ	Δ	○	○	○

Synchronization control enables an arbitrary control axis in the other part system to move in synchronization with the movement command assigned to an arbitrary control axis.
The direction in which the axis is to move synchronously can also be reversed using a parameter.

[Example of machining to synchronize Z2 axis to Z1 axis in the 1st part system (X1, Z1) and 2nd part system (Z2) (Lathe system)]



[Example of machining to synchronize X2 axis to X1 axis, Y2 axis to Y1 axis, and Z2 axis to Z1 axis in the 1st part system (X1, Y1, Z1) and 2nd part system (X2, Y2, Z2) (Machining center system)]



During synchronization, as the synchronized axis moves in synchronization with the reference axis regardless of automatic operation or manual operation, the movement command cannot be issued to the synchronized axis.

There are two types of control axis synchronization between part systems depending on the command method: Control axis synchronization between part systems I (G command) and control axis synchronization between part systems II (PLC signal). The method is chosen depending on the parameter setting.

■ Command format (Control axis synchronization between part systems I)

Synchronization start command

G125 Synchronized axis name = Reference axis name ;	
Synchronized axis name	: Command the axis name to be operated as synchronized axis under the control axis synchronization between part systems with two alphanumeric characters set to the parameter.
Reference axis name	: Command the axis name to be operated as reference axis under the control axis synchronization between part systems with two alphanumeric characters set to the parameter.

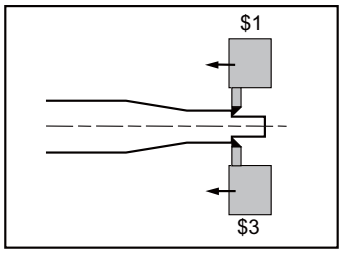
Synchronization end command

G125 Synchronized axis name ;	
Synchronized axis name	: Command the axis name to be operated as synchronized axis under the control axis synchronization between part systems with two alphanumeric characters set to the parameter.

12.1.8.6 Balance Cut

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	O	O	O	O	O	O	—

Multiple machining programs can be operated independently at same time for multi-axis and multi-part system mixed control CNC. This function enables the operation start timing synchronization between tool posts of two part systems.
When workpiece that is relatively long and thin is machined on a lathe, deflection may result, making it impossible for the workpiece to be machined with any accuracy. In cases like this, the deflection can be minimized by holding tools simultaneously from both sides of the workpiece and using them in synchronization to machine the workpiece (balance cutting). This method has an additional advantage: since the workpiece is machined by two tools, the machining time is reduced.
This function enables the movements belonging to two different part systems to be synchronized so that this kind of machining can easily be accomplished.



■ Command format

Balance cut command ON (modal)

!n L__ G15 ;
!n : Part system number for balance cut
Follows the parameter if the part system number is omitted.
L : Timing synchronization number (0 to 9999)

* !n LI can be omitted.

Balance cut command OFF (modal)

G14 ;

Note

(1) The G code varies depending on the G code list.

	G code list other than 6 or 7	G code list 6, 7
Balance cut command ON	G15	G68
Balance cut command OFF	G14	G69

G14 and G15 are modal commands. When the G15 command is assigned, the timing synchronization is being executed in all of the cutting feed command blocks until the G14 command is assigned or until the modal information is cleared by the reset signal. The timing synchronization is possible in all blocks by the parameter.

Part system 1 program

```
T 0 1 01;
G00  X_  Z_;
!3G15;
G01  Z_  F0. 4;
      :
```

Part system 3 program

```
T 0 1 0 2;
G00  X_  Z_;
!1G15
G01  Z_  F0. 4;
      :
```

Whereas synchronization is possible only with the next block when using the code "!" of timing synchronization between part systems, the balance cutting function provides synchronization (at the block start timing) with multiple consecutive blocks.

12.1.8.7 Common Memory for Part Systems

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	O	O	O	O	O	O	—

For a machine with multiple part systems, the common variables and tool compensation memory which exist for each part system can be made common for all part systems by setting the parameters.

■ Common variable

In the multi-part system, normally the common variables #100 to #199 are used for each part system, and variables #500 to #999 are common for the part systems. When this function is valid, the common variables #500 to #999 also become variables for each part system.
To use part or all of the common variables #100 to #199 and #500 to #999 commonly for all part systems, set the number of parameters to be used commonly for the part systems in variables #100 to #199 and #500 to #999.
As for the variables designated to be common to part systems, the data for part system 1 is used.

■ Tool compensation memory

When this specification is valid, the tool compensation memory used for operation in the 2nd or following part system changes to be used with the part system 1 data. Thus, the 2nd part system and following data is not used when the common data is used for the part systems.
This function is only for the lathe system and supported up to 2 part systems.

Note

(1) The setting (such as number of axes and axis names) for the tool compensation amount of part system 1 and part system 2 must be the same. If the settings differ, the setting of part system 1 will be applied.

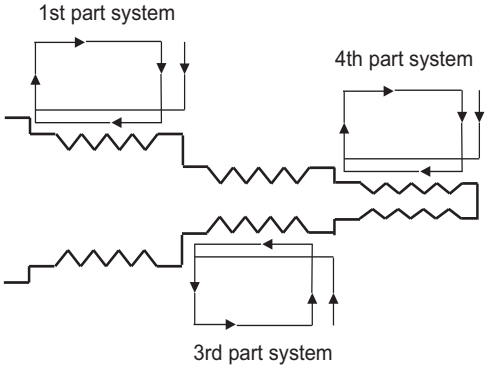
12.1.8.8 Multi-Part System Simultaneous Thread Cutting

Multi-part system simultaneous thread cutting allows different part systems to perform thread cutting simultaneously on one spindle.

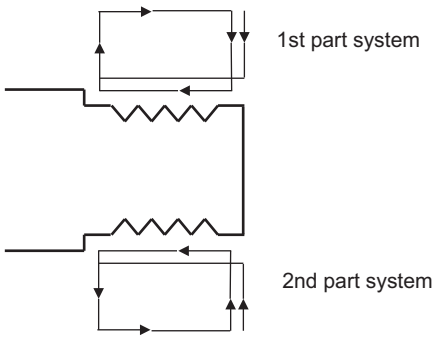
Multi-part system simultaneous thread cutting has two commands; the command (G76.1) for simultaneously cutting threads in multiple places, which is known as "multi-part system simultaneous thread cutting cycle I", and the command (G76.2) for simultaneously cutting a thread by two part systems, which is known as "two-part system simultaneous thread cutting cycle II".

Chamfering is available for multi-part system simultaneous thread cutting cycle I and two-part system simultaneous thread cutting cycle II.

Multi-part system simultaneous thread cutting cycle I (G76.1)



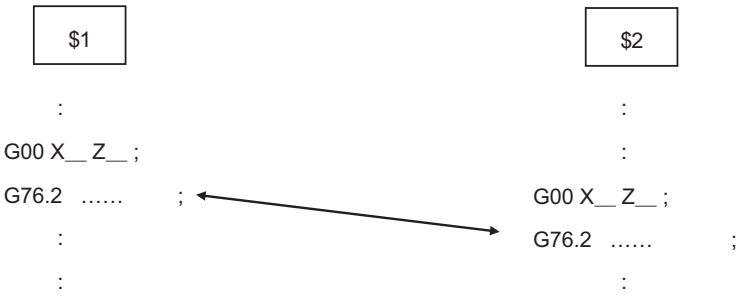
2-Part System Simultaneous Thread-Cutting Cycle II (G76.2)

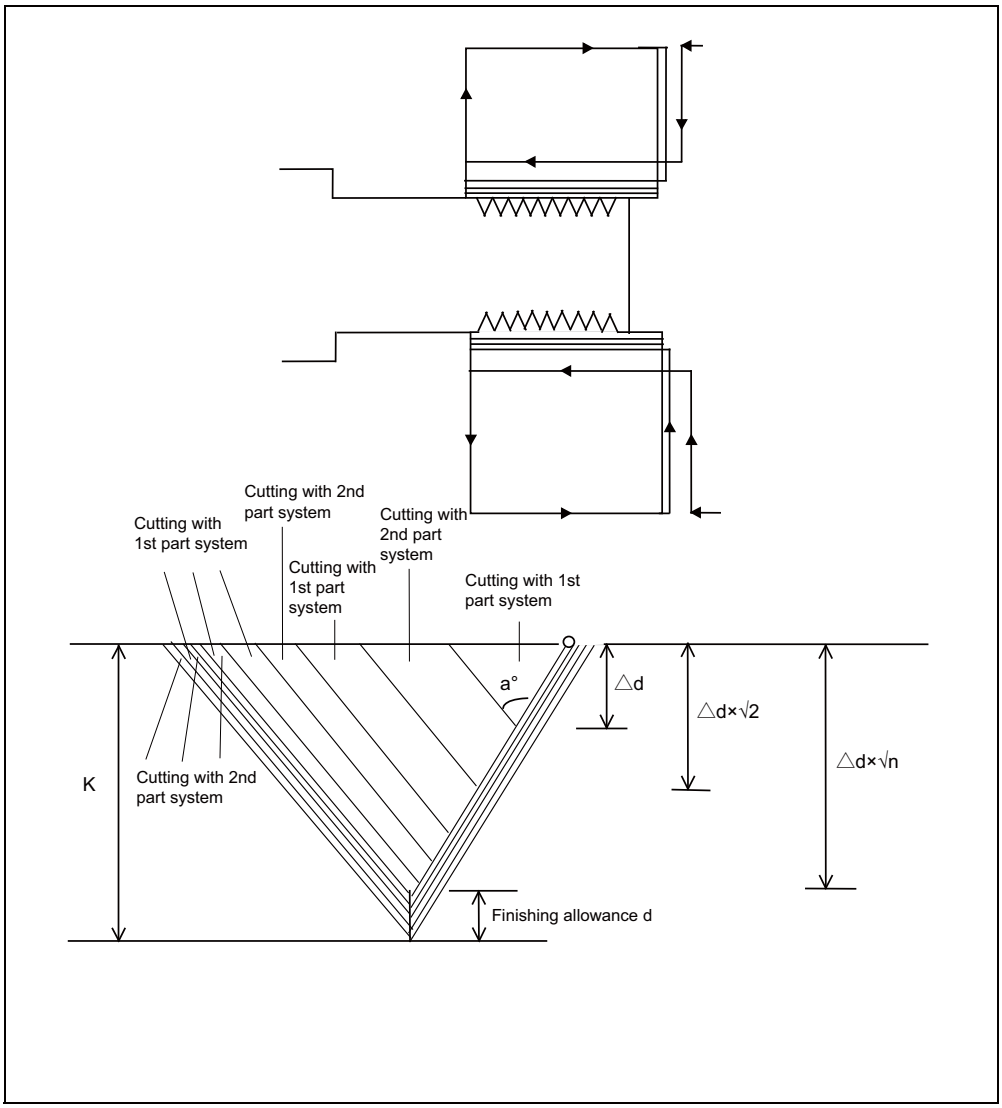


12.1.8.8.1 Two-Part System Simultaneous Thread Cutting

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	Δ	Δ	Δ	Δ	○	○	—

When G76.2 is issued in the 1st and 2nd part systems, waiting is done until G76.2 is issued in each part system.
The thread cutting cycle starts when the commands are aligned properly.
G76.2 assumes the same thread cutting, and deeply cuts in with the cutting amount using 1st part system and 2nd part system alternately.





■ Command format

G76.2 X/U__ Z/W__ R__ P__ Q__ A__ F__ ;	
X/U	: X-axis end point coordinates of thread part
Z/W	: Z-axis end point coordinates of thread section
R	: Taper height component at thread section
P	: Thread height
Q	: Cut depth
A	: Thread cutting start shift angle
F	: Thread lead

Note

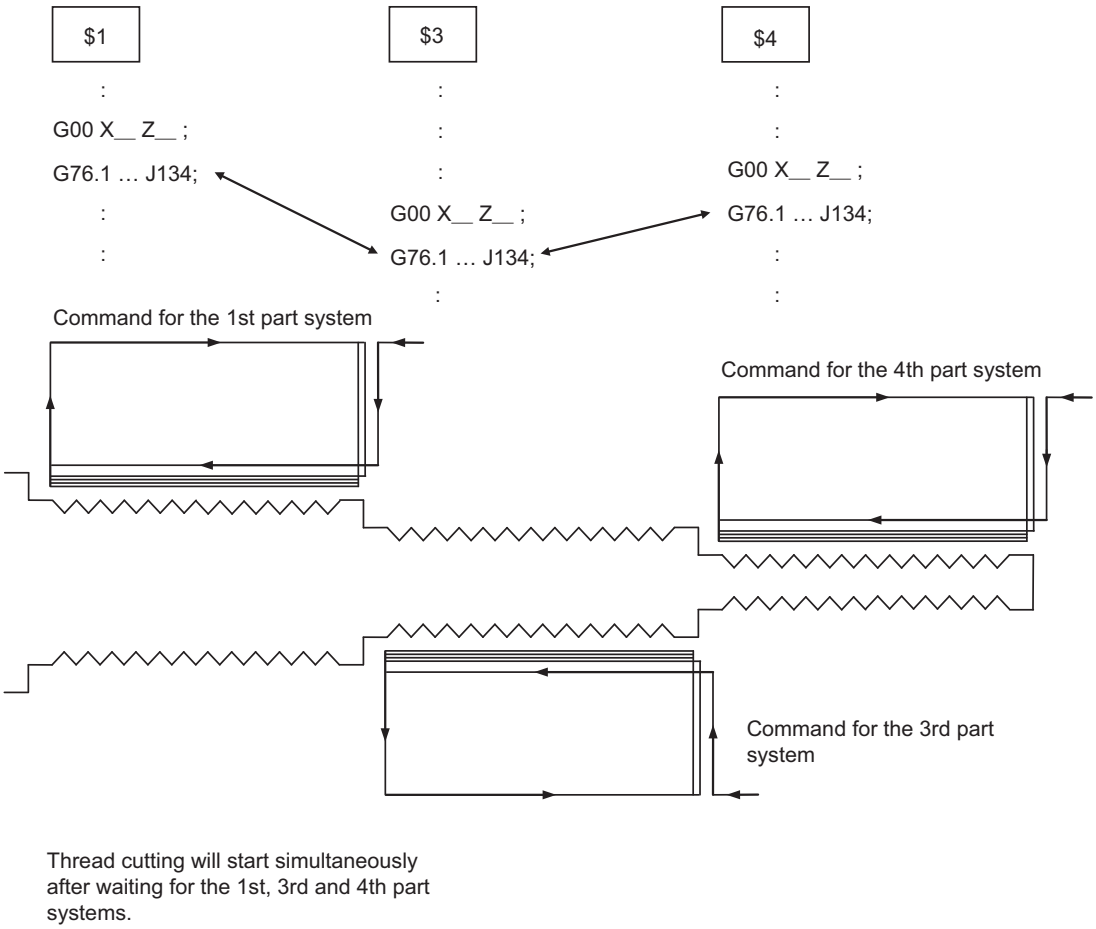
(1) Two-part system simultaneous thread cutting cycle II (G76.2) can be commanded in part system 1 or 2 only. A program error will occur when issuing the command in the other part systems.

12.1.8.8.2 Multi-Part System Simultaneous Thread Cutting

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	Δ	Δ	Δ	Δ	○	○	—

If G76.1 is given to multiple part systems, each part system waits for G76.1 to be commanded.
The thread cutting cycle starts when the G76.1 commands are aligned properly.

[When "J134" is commanded in $1, $3 and $4]



■ Command format

G76.1 X/U__ Z/W__ R__ P__ Q__ J__ F__ ;	
X/U	: X-axis end point coordinates of thread part
Z/W	: Z-axis end point coordinates of thread section
R	: Taper height component at thread section
P	: Thread height
Q	: Cut depth
J	: Part system for simultaneous thread cutting Ones digit: Part system number for the 1st set Tens digit: Part system number for the 2nd set : Ten-millions digit: Part system number for the 8th set
F	: Thread lead

12.1.8.9 Multi-Part System Program Management

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	—
L	○	○	○	○	○	○	○

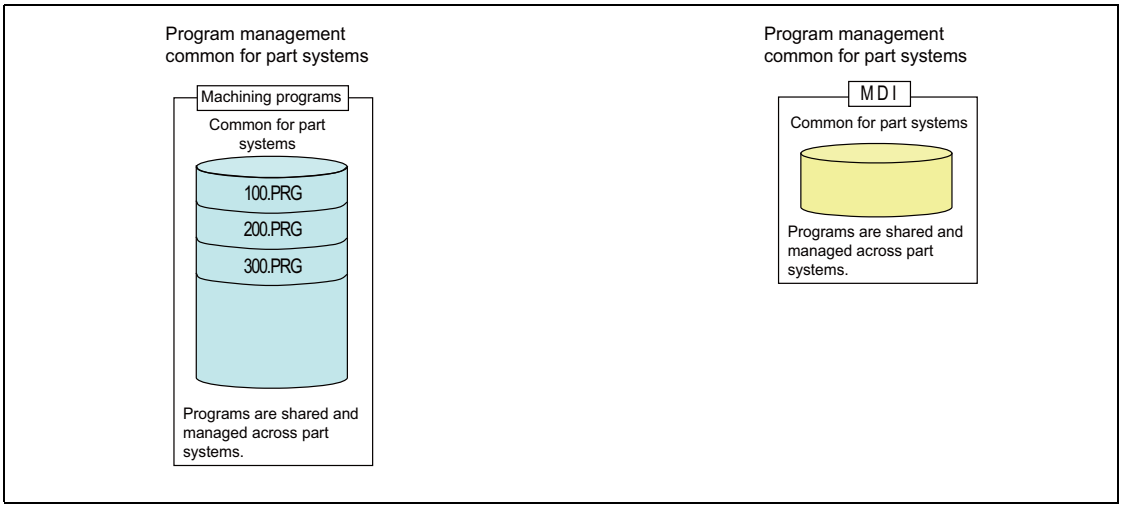
In a system composed of multiple part systems, this function manages the program storage method, program search method and program input/output.

- This facilitates the management of such machining programs that run simultaneously in multiple part systems.
- This enables separate programs that run simultaneously in multiple part systems to be handled under one common name.
- The parameter enables or disables this function.
- NC memory (*1) or NC memory 2 is the only device compatible with this function. (*2)

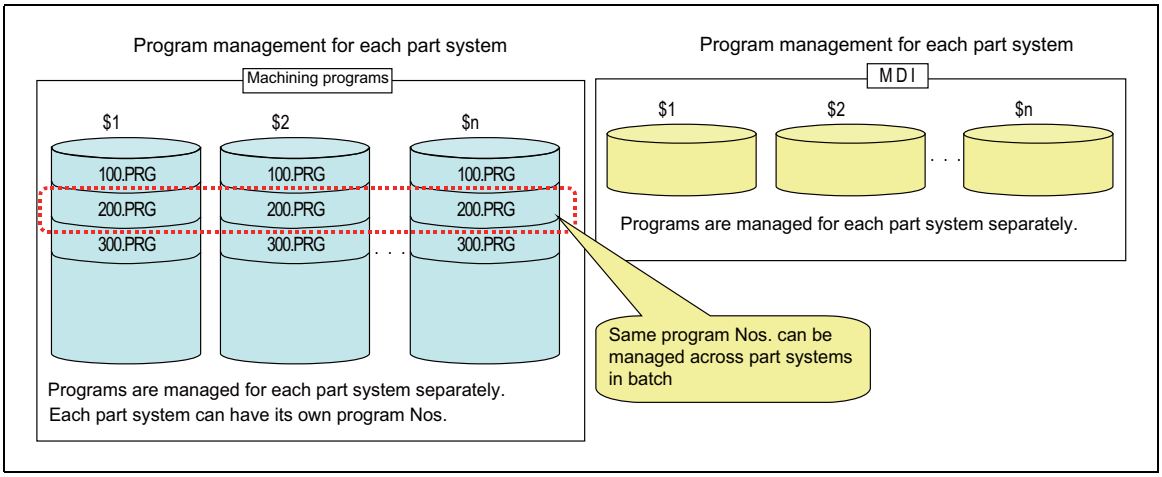
(*1) MDI and machine tool builder's macro are included.

(*2) NC memory is a different device from NC memory 2, therefore the program on each device is handled separately even when the program number or program name is the same as that of the other device.

[Multi-part system program management INVALID]



[Multi-part system program management VALID]



$1: Part system 1
$2: Part system 2
$n: Part system n

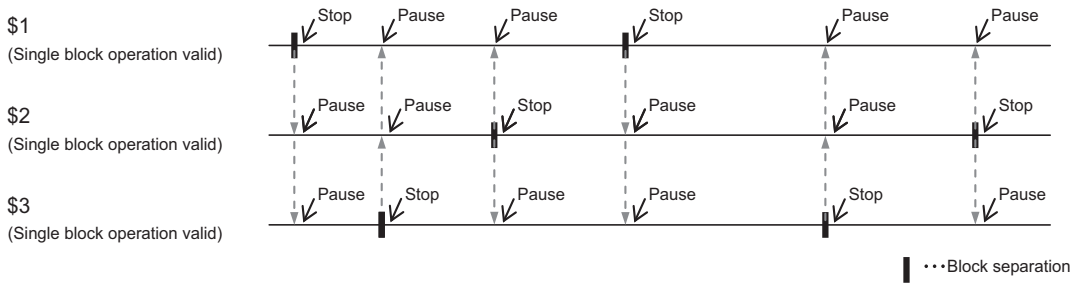
12.1.8.10 Synchronization Between Part Systems

12.1.8.10.1 Single Block Between Part Systems

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	Δ	Δ	Δ	Δ	○	○	○

Single block operation between part systems is the function for executing single block operation while maintaining the synchronization among the part systems when two or more part systems are operated in the multi-part system. When one part system has been stopped by single block stop, the other part systems pause in the cycle operation.
In the normal single block, the synchronization between the part systems differs from the synchronization during the continuous operation because the other part systems continue the operation without stop even when a part system is stopped by single block stop, and the tool interference could occur in some cases. The synchronization between part systems can be maintained the same as the continuous operation by the single block operation between part systems.
Valid/invalid can be switched with the PLC signal.

[3 part systems lathe]



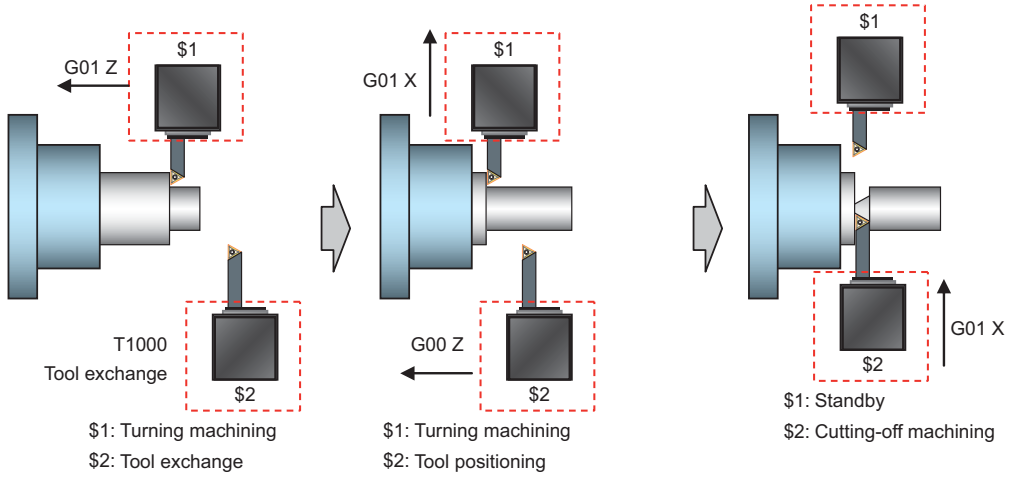
Stop: Single block stop
Pause: Cycle operation pause
$1: Part system 1
$2: Part system 2
$3: Part system 3

12.1.8.10.2 Dwell/Miscellaneous Function Time Override

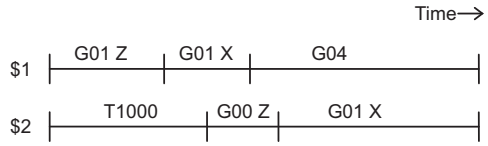
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	Δ	Δ	Δ	Δ	○	○	○

Override can be applied to dwell time and miscellaneous function finish wait time of all part systems. The synchronization between part systems can be maintained when the multiple machining programs are operated with override in the multi-axis and multi-part system mixed control CNC.
If a machining program that performs cutting-off machining in part system 2 ($2) after the completion of turning machining in part system 1 ($1) is executed with override when this function is invalid, part system 1 and part system 2 will be out of synchronization and the machining may not be operated properly.

The differences on the start timing for each command are shown in (1) to (3) when executing the machining with the part system 1 ($1) and part system 2 ($2) as shown in the figure below.

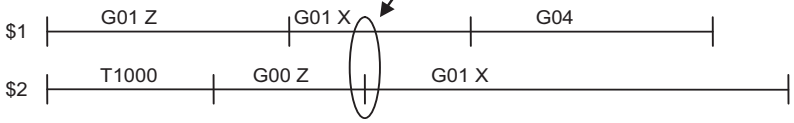


(1) Override 100%



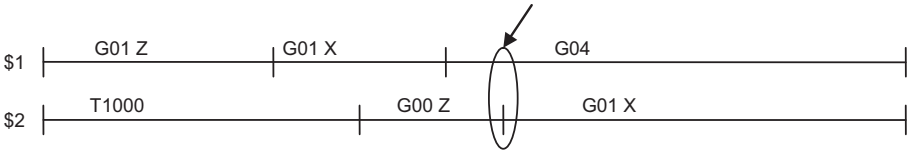
(2) When this function is invalid
Override 50%

Only feed time doubled in the operation with override 50%.
Part systems are out of synchronization and the cutting-off machining starts during turning machining.



(3) When this function is valid
Override 50%

Feed time, dwell time and miscellaneous function finish wait time double in the operation with override 50%.
The synchronization between part systems are maintained and the cutting-off machining stars after the completion of the turning machining.

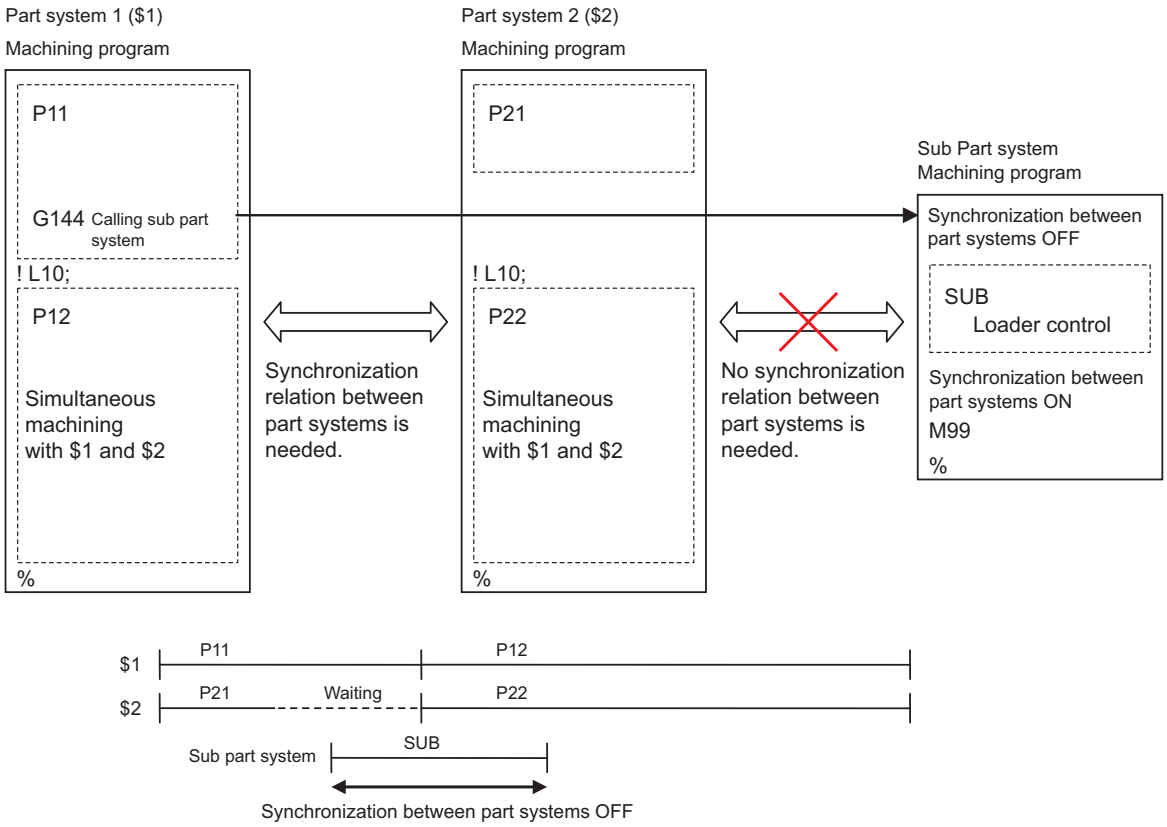


12.1.8.10.3 Synchronization Between Part Systems OFF

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	Δ	Δ	Δ	Δ	○	○	○

Synchronization between part systems and feedrate change are turned OFF in a part of a machining program to eliminate a synchronization relation between part systems by single block operation with part systems synchronized or variation of a machining program feedrate by dry run. This function is effective mainly in blocking the cycle operation pause or feedrate variation in only some of part systems when the sub part system control II function is being used. When loader control and ATC control which are operated independently of machining are performed with a sub program or sub part system control, this function enables a operation with synchronization between part systems in machining part systems and an independent operation in sub part systems.

ON/OFF can be switched with the system variable or the PLC signal.



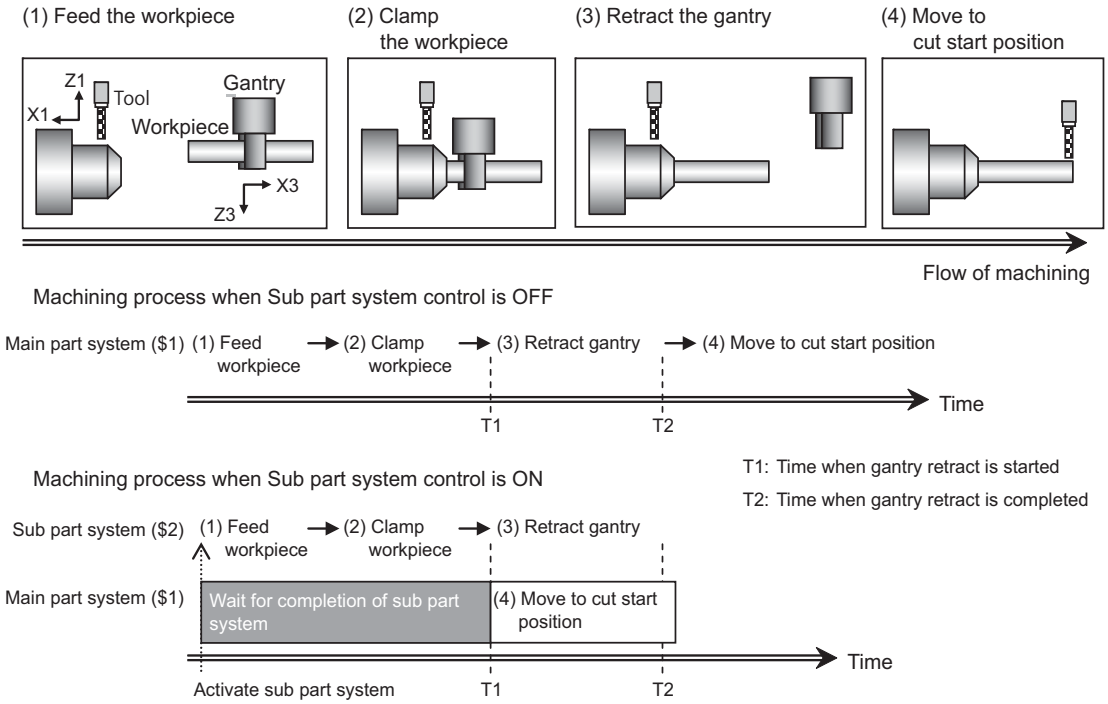
The behaviors with synchronization between part systems OFF is as follows:

Single block function between part systems	• The pause is not made from other part systems. • The other part systems do not pause with block stop.
Dwell/miscellaneous function time override	• Override is invalid.

12.1.8.11 Sub Part System Control I

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

This function activates and operates any non-operating part system (sub part system) in the multi-part system.
Sub part system control I can be used in the same manner as calling subprogram in a non-operating part system. An auxiliary axis machining program can be controlled in the sub part system by commanding Sub part system control I (G122) from the main part system.
In the usage example below, the tool positioning starts to the machining start point at the same time (time T1) as the start of gantry retract by using Sub part system control I (G145) in the flow from feeding the workpiece to moving to cut start position in order to reduce the cycle time.
Select whether main part system or sub part system for each part system in Sub part system control I. When using a part system as sub part system, by setting the operation mode to "Sub part system I operation mode" with the PLC signal and commanding Sub part system control I (G122) from an operating part system, it is possible to activate the part system in the sub part system I operation mode as sub part system.



■ Command format

Activate sub part system

Command an activation of sub part systems by Sub part system control I with G122.

G122 A__ P__ Q__ K__ D__ B__ H__ Argument;	
Or, G122 <File name> P__ Q__ K__D__ B__ H__ Argument;	
A	: Program No. (1 to 99999999 or 100010000 to 199999998)
<File name>	: File name of program (up to 32 characters)
P	: Start sequence number (Head of program if omitted.)
Q	: End sequence number (To end of program (M99) if omitted.)
K	: Number of repetitions (1 to 9999)
D	: Synchronization control (0/1)
B	: Sub part system identification number (1 to 7)
H	: Sub part system reset type (0/1)
Argument	: Argument of sub part system local variable (Setting rage of local variable (decimal point command is valid))

Complete sub part system

Command M99 in sub part system to complete an operation of sub part system.

```
M99 ;
```

Cancel the standby status for completion of sub part system

When a sub part system is activated in the completion wait method (D0 command), it is possible to cancel the standby status for completion of sub part system in a calling part system by commanding G145 in the sub part system program. G145 is ignored in a sub part system activated in the parallel processing method (D1 command).

```
G145 ;
```

Note

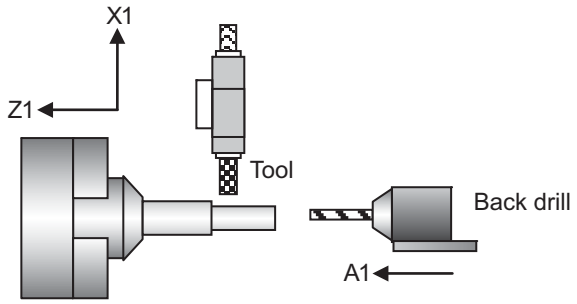
(1) For G code list 6 and 7, replace G122 with G153 and G145 with G155.

12.1.8.12 Sub Part System Control II

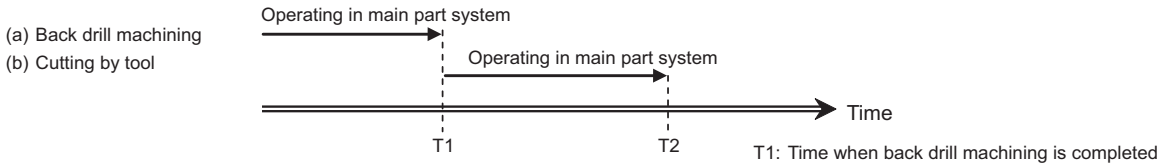
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	Δ	Δ	Δ	Δ	○	○	—

This function activates and operates any non-operating part system (sub part system) in the multi-part system.
Using sub part systems enables parallel operation between an operating program in main part system and a program called with Sub part system control II (G144).
Sub part systems does not have any controllable axes soon after the activation. When an axis is needed to be controlled in sub part system, command the arbitrary axis exchange control (G140) to give sub part system authorization to control the axis.

In the machine configuration below, when performing the back drill machining (machining (a)) and the cutting by tool (machining (b)), machining (a) and (b) are operated in order only with main part system, however machining (a) and (b) can be operated simultaneously by using the sub part system in addition to the main part system. This results in a shorter cycle time. In the usage example below, T2 (time when machining (b) is completed) is shorter.

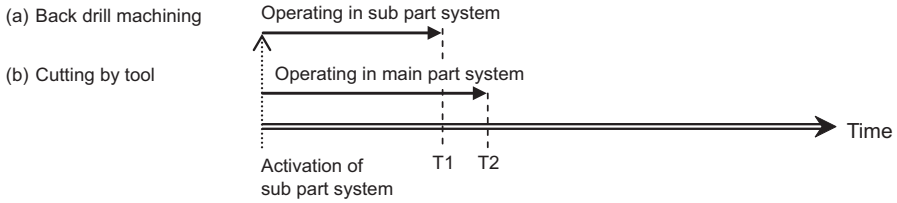


Machining process when Sub part system control is OFF



T1: Time when back drill machining is completed
T2: Time when cutting by the tool is completed

Machining process when Sub part system control is ON



■ Command format

Activate sub part system

Command an activation of sub part systems by Sub part system control II with G144.

G144 A__ P__ Q__ K__ D__ B__ H__ Argument;	
or, G144 <File name> P__ Q__ K__D__ B__ H__Argument;	
A	: Program No. (1 to 99999999 or 100010000 to 199999998)
<File name>	: File name of program (up to 32 characters)
P	: Start sequence number (Head of program if omitted.)
Q	: End sequence number (To end of program (M99) if omitted.)
K	: Number of repetitions (1 to 9999)
D	: Synchronization control (0/1)
B	: Sub part system identification number (1 to 9999)
H	: Sub part system designation (1 to number of part system that can be used)
Argument	: Argument of sub part system local variable (Setting rage of local variable (decimal point command is valid))

Complete sub part system

Command M99 in sub part system to complete an operation of sub part system.

```
M99 ;
```

Cancel the standby status for completion of sub part system

When a sub part system is activated in the completion wait method (D0 command), it is possible to cancel the standby status for completion of sub part system in a calling part system by commanding G145 in the sub part system program. G145 is ignored in a sub part system activated in the parallel control method (D1 command).

```
G145 ;
```

Note

(1) For G code list 6 and 7, replace G144 with G154 and G145 with G155.

12.1.8.13 Loader Control Part System

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	—	—	—
L	—	—	—	—	—	—	—

This function operates a part system which is dedicated to a carrier loader (hereinafter referred to as "loader") in the machining center system. This function can be used to isolate the loader system from other controls, allowing your machine to operate in automatic operation with CNC machining program.

Compared to when a loader is operated in a workpiece machining part system, the cycle time can be improved because the loader-specific process (such as placing the next workpiece and carrying the machined workpiece) is performed independently while a workpiece is being machined.

12.1.9 Data Input/Output by Program

12.1.9.1 Parameter Input by Program

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

The parameters set from the setting and display unit can be changed in the machining programs.
The command format is as follows.

G10 L70;	**Data setting start command**
P__ S__ A__ H □ __ ;	**Bit parameter**
P__ S__ A__ D__ ;	**Numerical value parameter**
P__ S__ A__ <character string> ;	**Character string parameter**

G11 ;	**Data setting end command**

P	Parameter No.
S	Part system No.
A	Axis No.
H	Bit type data
D	Numeric type data
<character string>	Character string data

Note

(1) The sequence of addresses in a block must be as shown above.
When an address is commanded two or more times, the last command will be valid.
(2) The part system No. is set in the following manner: "1" for 1st part system, "2" for 2nd part system, and so forth.
If the address S is omitted, the part system of the executing program will be applied.
As for the parameters common to part systems, the command of part system No. will be ignored.
(3) The axis No. is set in the following manner: "1" for 1st axis, "2" for 2nd axis, and so forth.
If the address A is omitted, the 1st axis will be applied.
As for the parameters common to axes, the command of axis No. will be ignored.
(4) Address H is commanded with the combination of setting data (0 or 1) and the bit designation (□) (0 to 7).
(5) Only the decimal number can be commanded with the address D.
The value that is smaller than the input setting increment (#1003 iunit) will be round off to the nearest increment.
(6) The character string must be put in angled brackets "<" and ">".
If these angle brackets are not provided, the program error will occur.
Up to 63 characters can be set.
(7) G10L70 or G11 must be commanded alone in a block. A program error will occur if it is not commanded alone in a block.
(8) The settings of the parameters with (PR) in the parameter list will be enable after the power is turned OFF and ON.
Refer to the parameter list in your manual.

12.1.9.2 Compensation Data Input by Program

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

The value of the workpiece coordinate systems selected can be set or changed by program commands.
The tool offset amounts, which have been set from the setting and display unit, can be input by program commands.

Workpiece coordinate system offset input

[M system]
The position of the workpiece coordinate systems selected by the G54 to G59 commands can be set or changed by program commands.
During the absolute command (G90) mode, the commanded offset amount serves as the new offset, whereas during the incremental command (G91) mode, the currently set offset plus the commanded offset serves as the new offset.

G code	Function
G10 L2 P0	External workpiece coordinate system setting
G10 L2 P1	Workpiece coordinate system 1 setting (G54)
G10 L2 P2	Workpiece coordinate system 2 setting (G55)
G10 L2 P3	Workpiece coordinate system 3 setting (G56)
G10 L2 P4	Workpiece coordinate system 4 setting (G57)
G10 L2 P5	Workpiece coordinate system 5 setting (G58)
G10 L2 P6	Workpiece coordinate system 6 setting (G59)
G10 L20 Pn (n=1 to 300)	Extended workpiece coordinate system setting (G54.1 P1 to P300)

■ Command format

G10 L2(L20) P__ X__ Y__ Z__ ;
G10 L2(L20) : Workpiece coordinate system setting command
P : Workpiece coordinate system No.
X, Y, Z : Setting value

Note

(1) L2 can be omitted. If the command P is omitted, the specified offset is treated as that of the currently selected workpiece coordinate system.
(2) L20 is used to designate the extended workpiece coordinate system.

[L system]

When the command is given with absolute positions (X, Z and R), the offset is updated with the values. On the other hand, when the command is given with incremental positions (U, W and C), the currently set offset plus the commanded offset serves as the new offset.

G code	Function
G10 L2 P0	External workpiece coordinate system setting
G10 L2 P1	Workpiece coordinate system 1 setting (G54)
G10 L2 P2	Workpiece coordinate system 2 setting (G55)
G10 L2 P3	Workpiece coordinate system 3 setting (G56)
G10 L2 P4	Workpiece coordinate system 4 setting (G57)
G10 L2 P5	Workpiece coordinate system 5 setting (G58)
G10 L2 P6	Workpiece coordinate system 6 setting (G59)
G10 L20 Pn (n=1 to 48)	Extended workpiece coordinate system setting (G54.1 P1 to P48)

■ Command format

G10 L2(L20) P__ X__ (U__) Z__ (W__) ;	
G10 L2(L20)	: Workpiece coordinate system setting command
P	: Compensation No.
X,Z	: Compensation amount for each axis (absolute)
U,W	: Compensation amount for each axis (incremental)

Note

(1) L2 can be omitted. If the command P is omitted, the specified offset is treated as that of the currently selected workpiece coordinate system.

(2) L20 is used to designate the extended workpiece coordinate system.

Tool compensation input

The tool compensation amounts, which have been set from the setting and display unit, can be input by program commands. The command format differs between the [M system] and the [L system].

[M system]

Type I

G code	Function
G10 L10	Tool compensation amount

Type II

G code	Function
G10 L10	Tool length shape compensation amount
G10 L11	Tool length wear compensation amount
G10 L12	Tool radius shape compensation amount
G10 L13	Tool radius wear compensation amount

■ Command format

G10 L__ P__ R__ ;	
G10 L	: Command for setting compensation amount
P	: Compensation No.
R	: Compensation amount

[L system]

G code	Function
G10 L10	Tool length compensation amount
G10 L11	Tool wear compensation amount

■ Command format

G10 L10(L11) P__ X__ (U__) Z__ (W__) [Additional axis]__ [2nd additional axis]_R__ (C__) Q__ ;	
G10 L10(L11)	: Command for setting compensation amount
P	: Compensation No.
X,Z	: Compensation amount for each axis (absolute)
U,W	: Compensation amount for each axis (incremental)
Additional axis	: Additional axis compensation amount (absolute)
2nd additional axis	: 2nd additional axis compensation amount (absolute)
R	: Nose R compensation amount (absolute)
C	: Nose R compensation amount (incremental)
Q	: Hypothetical tool nose point

Note

(1) L11 can be omitted.

M system turning tool compensation input

By setting the parameter in M system, the compensation amount, Nose R compensation amount and tool nose point for three basic axes can be written with the following G10 format. In the absolute command (G90) mode, the commanded compensation amount is new compensation amount. In the incremental command (G91) mode, new compensation amount is obtained by adding the commanded compensation amount to the currently set compensation amount.

■ Command format

Shape compensation

G10 L12 P__ X__ Y__ Z__ R__ Q__ ;	
P	: Tool shape compensation number (1 to the number of tool offset sets)
X	: X axis compensation amount
Y	: Y axis compensation amount
Z	: Z axis compensation amount
R	: Tool nose radius compensation amount
Q	: Hypothetical tool nose point number

Wear compensation

G10 L13 P__ X__ Y__ Z__ R__ Q__ ;	
P	: Wear compensation number (1 to the number of tool offset sets)
X	: X axis compensation amount
Y	: Y axis compensation amount
Z	: Z axis compensation amount
R	: Tool nose radius compensation amount
Q	: Hypothetical tool nose point number

Cancelling the compensation input

Command format

G11 ;

12.1.9.3 Tool/Material Shape Input by Program

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

Tool shape data on the tool management screen (M system) and workpiece shape data (L system) of the 3D solid program check (hereinafter referred to as 3D check) can be set with the machining program. There is no need to input the tool/ workpiece shape on the screen for 3D check.

■ L100 command format

Tool shape data on the tool management screen can be set by the machining program with this function.

G10 L100;	**Data setting start command**

P__ T__ K__ D__ H__ I__ J__ C__ ;	**Data setting command**

G11 ;	**Data setting end command**

P	Data number (The maximum number varies depending on the number of tool management data sets.)
T	Tool number (0 to 99999999)
K	Type
D	Shape data 1 (decimal point allowed)
H	Shape data 2 (decimal point allowed)
I	Shape data 3 (decimal point allowed)
J	Shape data 4 (decimal point allowed)
C	Tool color

Note

(1) Omitted addresses cannot be set.
(2) Omitting address P or T causes the program error.
(3) On graphic check of M80V Series, tool shape data of tool management screen will be rewritten.
(4) On graphic check of M800V and M80VW Series, the specified data is reflected only in the drawing on the graphic check screen. Tool shape data will not be rewritten.
(5) This function can be used only with the M system.

■ L101 command format

Workpiece shape data can be set by the machining program for 3D solid program check with this function.

G10 L101;	**Data setting start command**

C__ R__ D__ E__ L__ A__ K__ W__ F__ P__ H__ I__ J__ ;	**Data setting command**

G11 ;	**Data setting end command**

C	Workpiece shape
R	Number of angles (3 to 99)
D	Outside diameter of workpiece (0.001 to 99999.999 mm, 0.0001 to 9999.9999 inch)
E	Inside diameter of workpiece (0.001 to 99999.999 mm, 0.0001 to 9999.9999 inch)
L	Workpiece length (0.001 to 99999.999 mm, 0.0001 to 9999.9999 inch)
A	Installation angle (0.000 to 359.999°)
K	Front face workpiece zero point position Z (-1 or 0.000 to 99999.999 mm, -1 or 0.0000 to 9999.9999 inch)
W	Back face workpiece zero point position Z (-1 or 0.000 to 99999.999 mm, -1 or 0.0000 to 9999.9999 inch)
F	Workpiece color
P	Machining surface color
H	Tapping/Thread machining surface color
I	Interference surface color
J	Cross-section color

Note

(1) Omitted addresses cannot be set.
(2) Omitting address C causes the program error.
(3) On graphic check of M80V Series, workpiece shape data of tool management screen will be rewritten.
(4) On graphic check of M800V and M80VW Series, the specified data is reflected only in the drawing on the graphic check screen. Workpiece shape data will not be rewritten.
(5) This function can be used only with the L system.

12.1.9.5 API Section and Sub-Section Nos. Input/Output by Program

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

NC internal data can be read/written by specifying the section number, sub-section number, part system number and axis number using system variables.

12.1.9.6 R-Navi Data Input by Program

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	—
L	—	—	—	—	—	—	—

This function enables the R-Navi setup parameters to be configured from a machining program.
After the parameters have been configured from the program, you can check the values or select the machining surface from the setup screen.

■ Command format

G10 L110 ;	**Workpiece data setting**
G10 L111 ;	**Machining surface data setting**

For details on R-Navi, refer to " 17.2.17 R-Navi".

12.1.10 Machining Modal

12.1.10.1 Tapping Mode

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

When tapping mode commands are issued, the NC system is set to the following internal control modes required for tapping.
- Cutting override is fixed at 100%.
- Deceleration commands at joints between blocks are invalid.
- Feed hold is invalid.
- Single block is invalid.
- "In tapping mode" signal is output.

G code	Function
G63	Tapping mode ON

The tapping mode command will be canceled with the following commands:
- Exact stop check mode (G61)
- Automatic corner override (G62)
- Cutting mode (G64)
- High-accuracy control mode command (G61.1/G08P1) [M system]

The machine is in the cutting mode status when its power is turned ON.

12.1.10.2 Cutting Mode

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

When a cutting mode command is issued, the NC system is set to the cutting mode that enables smooth cutting surface to be achieved. In this mode, the next block is executed continuously without the machine having to decelerate and stop between the cutting feed blocks: this is the opposite of what happens in the exact stop check mode (G61).

G code	Function
G64	Cutting mode ON

The cutting mode command will be canceled with the following commands:
- Exact stop check mode (G61)
- Automatic corner override (G62)
- Tapping mode (G63)
- High-accuracy control mode command (G61.1) [M system]

The machine is in the cutting mode status when its power is turned ON.

12.1.11 High-Speed Parts Machining

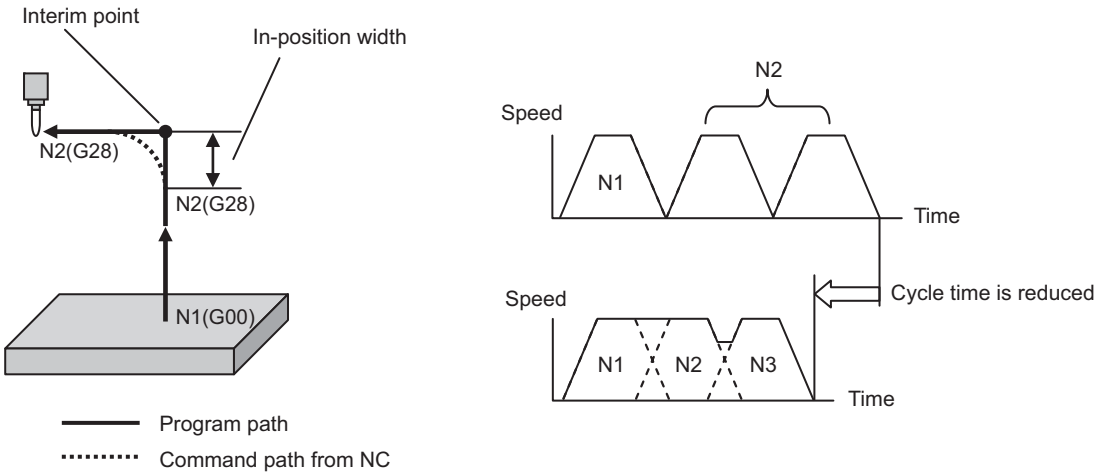
12.1.11.1 Rapid Traverse Block Overlap

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

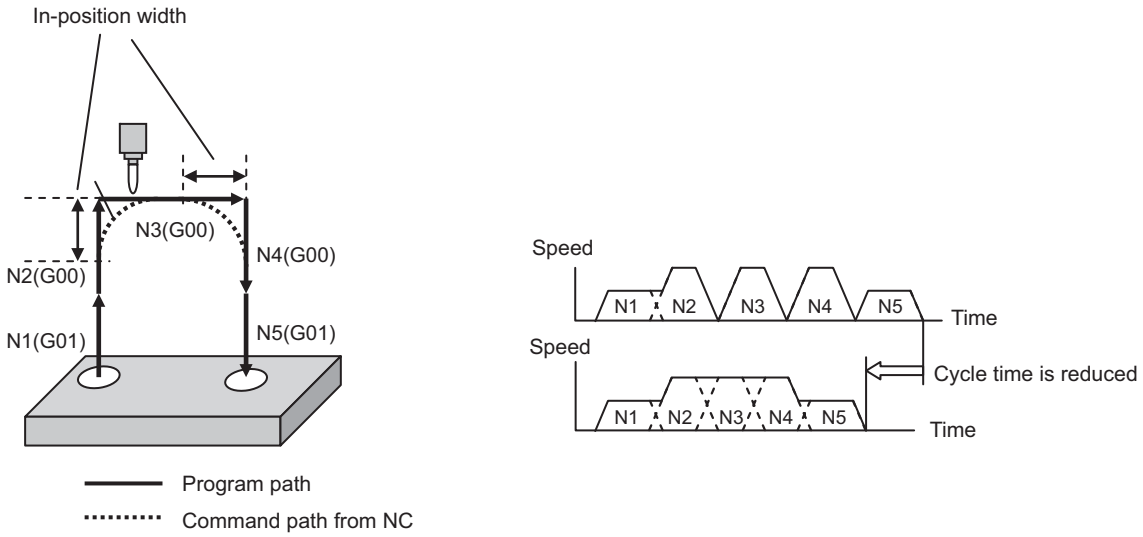
This function enables the next block to start (overlap) without waiting for positioning (G00) or reference position return (G28/G30).
Consequently, cycle time of machining including operation of positioning (G00) or reference position return (G28/G30) can be reduced.
Adjust the overlap amount with command by machining program or with the parameter, and specify it as in-position width for rapid traverse block overlap.
Also, the operation does not decelerate between blocks if the movement command continues in same direction.
The overlap is also valid when the block after G00 is G01 instead of G00 or G28/G30.
It is invalid when G00 or G28/G30 is commanded after G28 continuously.
This function has two modes. The modes can be selected with the parameter.

Mode 1: The cycle time can be reduced in normal acceleration/deceleration control.
Mode 2: It is also effective to reduce the cycle time in pre-interpolation acceleration/deceleration control. (Recommended)

[Example of application of rapid traverse block overlap in tool exchange operation]



[Example of application of rapid traverse block overlap in continuous drilling operation]



■ Command format

The command format to switch the rapid traverse block overlap function for G00 to valid/invalid is as follows:

G0.5 P1 J__ K__ ;	**Rapid traverse block overlap for G00 is valid**

J	Liner axis in-position width (0.000 to 1000.000 (mm))
K	Rotary axis in-position width (0.000 to 1000.000 (°))

G0.5 P0 ;	**Rapid traverse block overlap for G00 is invalid**

12.2 Machining Accuracy Support Functions

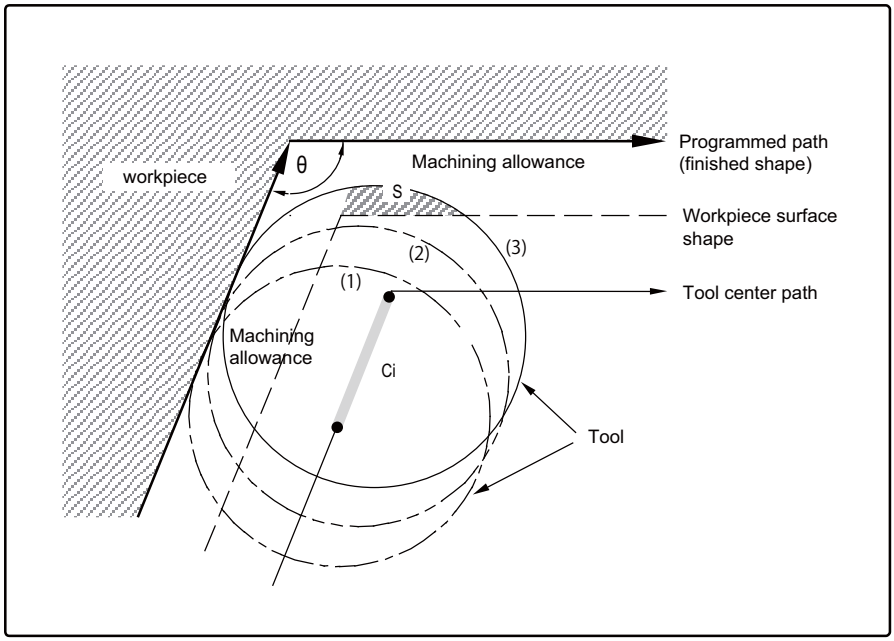
12.2.1 Automatic Corner Override

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

When cutting with tool radius compensation, this function applies feedrate override automatically to decrease cutting load at the inside corner cutting or the inside cutting of corner R.
There are automatic corner override (G62) and inside arc override as the automatic corner override.

Automatic corner override (G62) is valid until the tool radius compensation cancel (G40), exact stop check mode (G61), high-accuracy control mode (G61.1), tapping mode (G63), or cutting mode (G64) command is issued.
Inside arc override is valid without automatic corner override (G62) during tool radius compensation mode (G41/G42). This can also be enabled with the parameter setting when the tool radius compensation mode (G41/G42) and the automatic corner override mode (G62) are active.

[Automatic corner override (G62) inside corner]



θ : Maximum angle at inside corner
Ci : Deceleration range (IN)

Operation

- When G62 command is not given:
  When the tool moves in the order of (1) -> (2) -> (3) in the figure above, the machining allowance at (3) is larger than that at (2) by an amount equivalent to the area of shaded section S and so the tool load increases.
- When G62 command is given :
  When the inside corner angle θ in the figure above is less than the angle set in the parameter, the override set into the parameter is automatically applied in the deceleration range Ci.

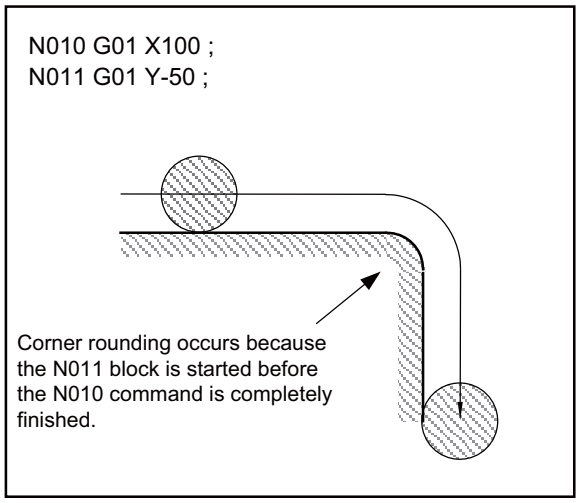
■ Command format

G62; Automatic corner override

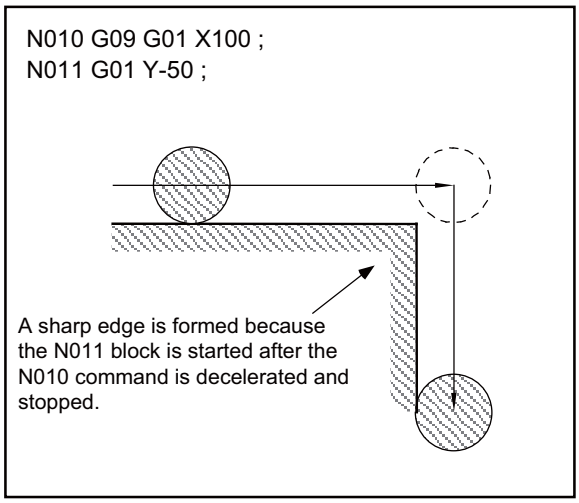
12.2.2 Deceleration Check

This function leads the machine to decelerate and stop at the join between axis movement blocks before executing the next block. This is effective to alleviate the machine shock and prevent the corner rounding when the feed rate of the control axis changes suddenly.
There are three methods for deceleration check: command deceleration check method, smoothing check method and in-position check method.

Without deceleration check



With deceleration check



The conditions for executing deceleration check are described below.

[Deceleration check in the rapid traverse mode]

In the rapid traverse mode, the deceleration check is always performed when block movement is completed before executing the next block.

[Deceleration check in the cutting feed mode]

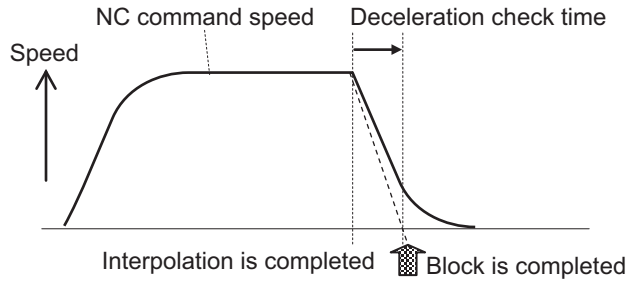
In the cutting feed mode, the deceleration check is performed at the end of block when any of the conditions below is applicable before executing the next block.

- When the G61 (exact stop check mode) command has been selected.
- When the G09 (exact stop check) is issued in the same block.
- When the error detect switch (PLC signal) is ON.

The following deceleration check methods can be selected with the parameter to each feed command during rapid traverse command and cutting feed command.

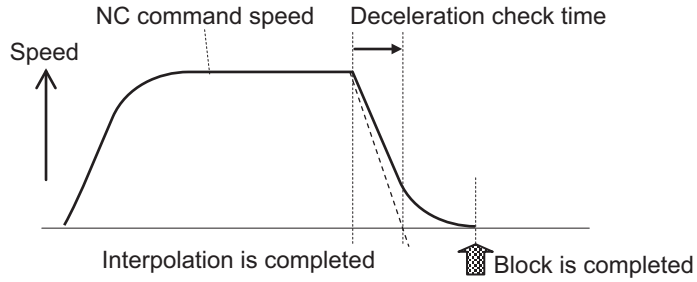
■ Command deceleration check method

Deceleration is completed after the deceleration check time(*1) has passed after the interpolation.



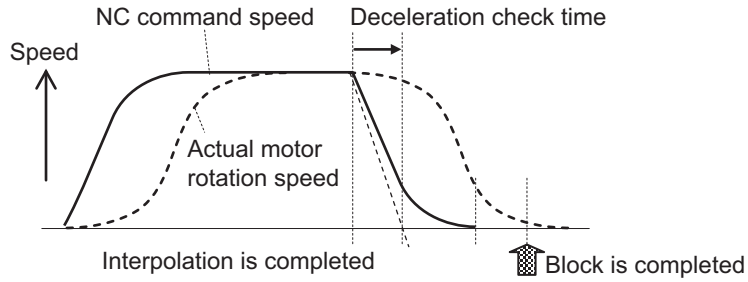
■ Smoothing check method

Deceleration is completed after the deceleration check time(*1) has passed after the interpolation and all axes smoothing has become zero.



■ In-position check method

Deceleration is completed after the deceleration check time(*1) had passed after the interpolation, all axes smoothing has become zero and all axes have become in-position.



(*1) Deceleration check time: automatically calculated from the acceleration/deceleration mode and time constant.

12.2.2.1 Exact Stop Check Mode

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

A deceleration check is performed when the G61 (exact stop check mode) command has been selected.
G61 is a modal command. The deceleration is applied at the end points of each block to all the cutting commands (G01 to G03) subsequent to G61 and the in-position status is checked.
For details on the deceleration check, refer to " 12.2.2 Deceleration Check".

12.2.2.2 Exact Stop Check

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

A deceleration check is performed when the G09 (exact stop check) command has been designated in the same block.
The G09 command should be issued in the same block as the cutting command. It is an non-modal command.
For details on the deceleration check, refer to " 12.2.2 Deceleration Check".

12.2.2.3 Error Detection

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

To prevent rounding of a corner during cutting feed, the operation can be changed by turning an external signal switch ON so that the axis decelerates and stops once at the end of the block and then the next block is executed.
The deceleration stop at the end of the cutting feed block can also be commanded with a G code.
For details on the deceleration check, refer to " 12.2.2 Deceleration Check".

12.2.2.4 Programmable In-Position Check

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

,I address is used to designate the in-position width for a linear interpolation command from the machining program.
The in-position width designated with a linear interpolation command is valid only in cases when the deceleration check is performed, such as:

- When the error detect switch is ON.
- When the G09 (exact stop check) is issued in the same block.
- When the G61 (exact stop check mode) command has been selected.

G01 Xx1 Zz1 Ff1 ,Ii1 ;

Xx1, Zz1	: Linear interpolation coordinate position of each axis
Ff1	: Feed rate
,Ii1	: In-position width

As with linear interpolation, ",I" address is used to designate the in-position width for a positioning command from the machining program.

G00 Xx1 Zz1 ,Ii1 ;

Xx1, Zz1	: Positioning coordinate position of each axis
,Ii1	: In-position width

[In-position check operation]
After it is verified that the position error of the block in which the deceleration check is performed by the positioning command (G00: rapid traverse) and the linear interpolation command (G01) is less than the in-position width of this command, the execution of the next block is commenced.

12.2.2.5 Automatic Error Detection

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	Δ	Δ	Δ	Δ	—	—	—

This function is effective to reduce the extension of cycle time for the cutting at the corner and realize the high edge accurate machining.

12.3 High-Speed and High-Accuracy Functions [kBPM: k Block per Minute]

12.3.1 High-Speed Machining Mode

12.3.1.1 High-Speed Machining Mode I (G05P1) Maximum [kBPM]

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ33.7	Δ33.7	Δ33.7	Δ33.7	○ 33.7	○ 33.7	○ 16.8
L	Δ33.7	Δ33.7	Δ33.7	Δ33.7	○ 33.7	○ 33.7	—

This function runs a machining program that approximates a free curve with micro segments at a high speed.
This is effective in increasing the speed of machining dies with a free curve.

■ Command format

G05 P1 ;	High-speed machining mode I start

G05 P0 ;	High-speed machining mode I cancel

In addition to the G05P0 command, the high-speed machining mode I is canceled with the following commands.
- High-speed machining mode II (G05P2)
- High-speed High-accuracy control I (G05.1Q1)
- High-speed High-accuracy control II (G05P10000)

■ Micro segment capacity

G1 block micro segment capacity (unit: kBPM)

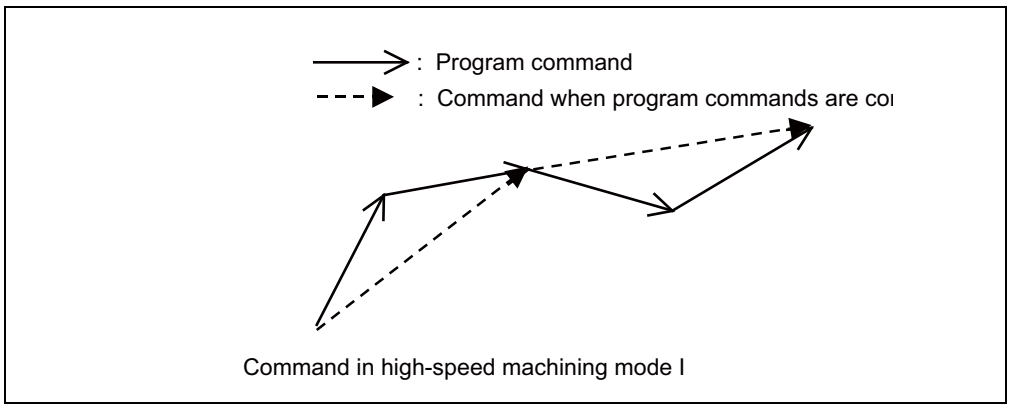
Mode	Command	Maximum feed rate when G1 block is executed
High-speed machining mode Ⅰ	G05 P1	33.7 (M800VW/M80VW/M800VS/M80V TypeA) 16.8 (M80V TypeB)

The above performance applies under the following conditions.

- 6-axis system (including spindle) or less
- 1-part system
- Three axes or less commanded simultaneously in G01
- Block containing only axis name and movement amount (Does not contain macro or variable command)
- During G61.1 high-accuracy control mode or during cutting mode (G64)

It may not be possible to attain the specified feed rate if deviated from the above conditions.

- During the high-speed machining mode I, the blocks are pre-read. If the length of the block is sufficiently short in respect to the command speed, and enough to be completed with one interpolation calculation, several blocks may be combined and converted into a block length that can be completed with one interpolation.



Command in high-speed machining mode I

- Override, maximum cutting speed clamp, single block operation, dry run, manual interruption, graphic trace and high-accuracy control mode are valid even during the high-speed machining mode I.

12.3.1.2 High-Speed Machining Mode II (G05P2) Maximum [kBPM]

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ168	Δ168	Δ168	Δ168	○ 101	○ 101	○ 67.5
L	Δ168	Δ168	Δ168	Δ168	○ 101	○ 101	—

This function runs a machining program that approximates a free curve with micro segments at a high speed.
Higher micro segment processing capacity can make the cutting speed faster, resulting in reducing the cycle time and improving the quality of the machining surface.
This function can be used simultaneously in up to two part systems.

■ Command format

G05 P2 ;	**High-speed machining mode II start**

G05 P0 ;	**High-speed machining mode II cancel**

In addition to the G05P0 command, the high-speed machining mode II is canceled with the following commands.
- High-speed machining mode I (G05P1)
- High-speed High-accuracy control I (G05.1Q1)
- High-speed High-accuracy control II (G05P10000)
- High-speed High-accuracy control II (G05P20000)

■ Micro segment capacity

The following is the micro segment capacity while high-speed machining mode II is valid.

Micro segment processing capacity (unit: kBPM)

Number of part systems/ Number of axes	Number of high-speed high-accuracy control-enabled part systems	M850V / M830V		M80VW / M80V Type-A		M80V Type-B	
		M	L	M	L	M	L
• 1-part system (8 axes or less)	1 part system	168 135 (*3) 101 (*1) 67.5 (*2)(*4)	168 135 (*3) 101 (*1)	101 67.5 (*1) 33.7 (*2)(*4)	101 67.5 (*1)	67.5	—
• 1-part system (9 axes or more) • 2-part system	1 part system	135 101 (*3) 67.5 (*1) 33.7 (*2)(*4)	135 101 (*3) 67.5 (*1)	101 67.5 (*1) 33.7 (*2)(*4)	101 67.5 (*1)	67.5	—
	2 part systems	67.5 33.7 (*2)	67.5	33.7	33.7	—	—
• 3-part system	1 part system	—	33.7	33.7	33.7	—	—
	2 part systems	—	33.7	33.7	33.7	—	—
• 4-part system (16 axes or less)	1 part system	33.7	33.7	33.7	33.7	—	—
	2 part systems	33.7	33.7	33.7	33.7	—	—
• 4-part system (17 axes or more) • 5-part system or more	1 part system	16.8	16.8	—	—	—	—
	2 part systems	16.8	16.8	—	—	—	—

(*1) This value indicates the micro segment processing capacity in the time constant expansion system or in the following modes.
- Tool center point control (G43.4/G43.5)
- Inclined surface machining (G68.2/G68.3)
- Workpiece installation error compensation (G54.4P1 to 7)
- 3-dimensional tool radius compensation (G41.2/G42.2)
- Spatial error compensation
- Variable-acceleration pre-interpolation acceleration/deceleration
- Spline interpolation 2
- 3D machine interference check
- Tool cutting point control (G43.8/G43.9)
- Real-time 3D machine interference check

(*2) This value indicates the micro segment processing capacity when Smooth fairing is ON.

(*3) This value indicates the micro segment processing capacity in tolerance control mode.

(*4) When Smooth fairing is ON and the control cycle is twice the standard cycle, a program error occurs.

Note

(1) The above performance applies under the following conditions.
- Three axes or less commanded simultaneously in G01
- Block containing only axis name and movement amount (Does not contain macro or variable command)
- During tool radius compensation OFF (G40)

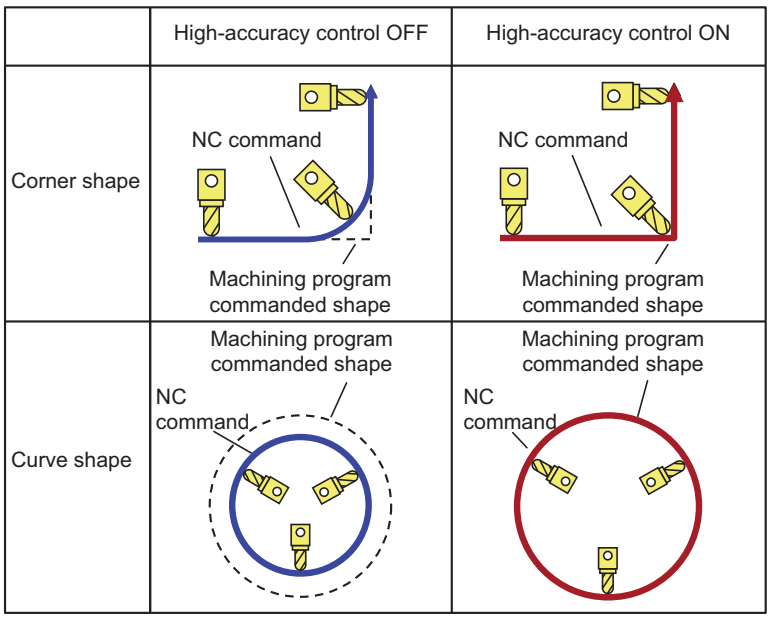
It may not be possible to attain the specified feed rate if deviated from the above conditions.

12.3.2 High-Accuracy Control

12.3.2.1 High-Accuracy Control (G61.1/G08)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

Machining errors caused by delays in control systems can be inhibited. This function is useful for machining which needs to make an edge at a corner or reduce an error from an inner route of curved shape. In high-accuracy control, acceleration/deceleration is performed not to cause machining error by pre-reading blocks and acceleration/deceleration is automatically performed according to a machining shape so that the machining error is inhibited with minimizing an extension of machining time.



High-accuracy control is valid in High-accuracy control command (G08P1/G61.1), High-speed high-accuracy control I command (G05.1Q1), High-speed high-accuracy control II command (G05P10000), High-speed high-accuracy control III command (G05P20000) and Spline interpolation command (G61.2).

One or more of the functions in the table below must be ON. Program error occurs if the high-accuracy control is commanded with all of functions OFF status.

Function name	Details
High-accuracy control	This function enables High-accuracy control.
High-speed high-accuracy control I	This function enables High-speed high-accuracy control I.
High-speed high-accuracy control II	This function enables High-speed high-accuracy control II.
High-speed high-accuracy control III	This function enables High-speed high-accuracy control III.
Spline interpolation	This function enables Spline interpolation.

■ Command format

High-accuracy control ON

```
G61.1 ;
```

```
G08 P1;
```

High-accuracy control OFF

```
G08 PO ;
```

```
G command in G code group 13 expect G61.1
```

High-accuracy control can be cancelled with either command regardless of enabled high-accuracy control.

High-accuracy control inhibits machining errors with minimizing an extension of machining time using the following functions.

- Acceleration/deceleration before interpolation
- Optimum speed control
- Vector accuracy interpolation
- Feed forward control
- S-pattern filter control

Acceleration/deceleration before interpolation

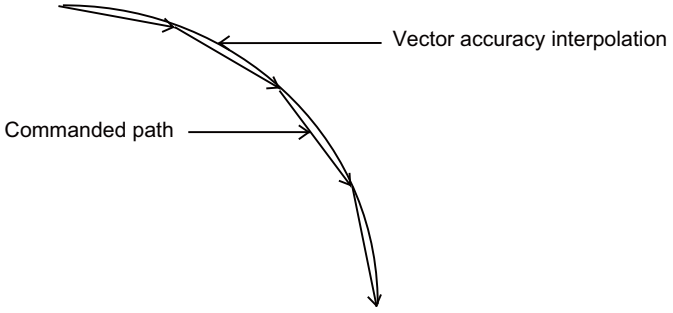
Velocity waveform is smoothed by performing acceleration/deceleration control on movement commands to inhibit shocks caused by rapid acceleration at the machine movement start/stop. However, if the high-accuracy control is OFF, a corner rounding may occur at the joint between blocks or a path error may occur for the commanded shape which has been described in machining program because the acceleration/deceleration process is performed after the interpolation.
If the high-accuracy control is ON, the acceleration/deceleration process is performed before the interpolation to solve the problems mentioned above. This acceleration/deceleration before interpolation enables machining with a faithful path to the commanded shape of machining program. Furthermore, the acceleration/deceleration time can be reduced because the constant-gradient acceleration/deceleration is performed for the acceleration/deceleration before interpolation.

Optimum speed control

When the moving direction is changed on the corner, arc, etc., acceleration corresponding to the amount of change and the feedrate is generated. When the acceleration is large, there is a possibility of machine vibration and it may remain stripes on the machining surface.
In the high-accuracy control mode, the deceleration control (optimum speed control) is performed to keep the generated acceleration under the allowance which has been designed with the parameter so that the problem mentioned above can be solved. The optimum speed control suppresses the machine vibration and enables highly accurate machining while minimizing the extension of cycle time.

Vector accuracy interpolation

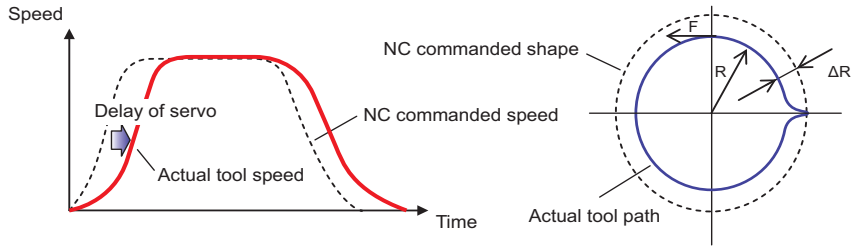
When a micro segment is commanded and the angle between the blocks is extremely small (when not using corner deceleration), interpolation can be carried out more smoothly using the vector accuracy interpolation.



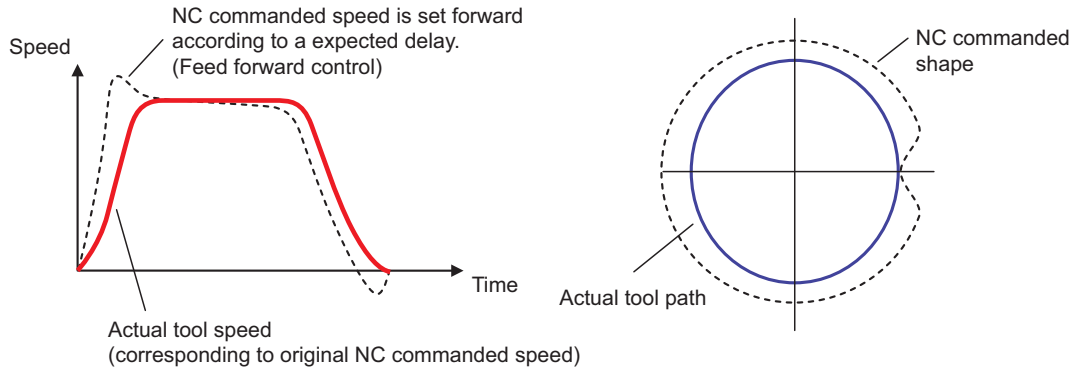
Feed forward control

This function reduces path errors caused by delay of servo systems. Path errors caused by acceleration/deceleration of NC can be eliminated by acceleration/deceleration before interpolation, however errors caused by delay of servo systems cannot be eliminated by acceleration/deceleration before interpolation. Therefore, when the arc shape of radius R (mm) is machined at speed F (mm/min) as the figure (a) below, for instance, the lag time occurs between the NC commanded speed and the actual tool speed in amount of the servo system time constant and the path error ΔR (mm) occurs. Feed forward control generates the command value taking the delay of servo systems as shown in figure (b) below so that the path error caused by delay of servo systems can be inhibited.

■ NC command and actual tool movement during Feed forward control OFF



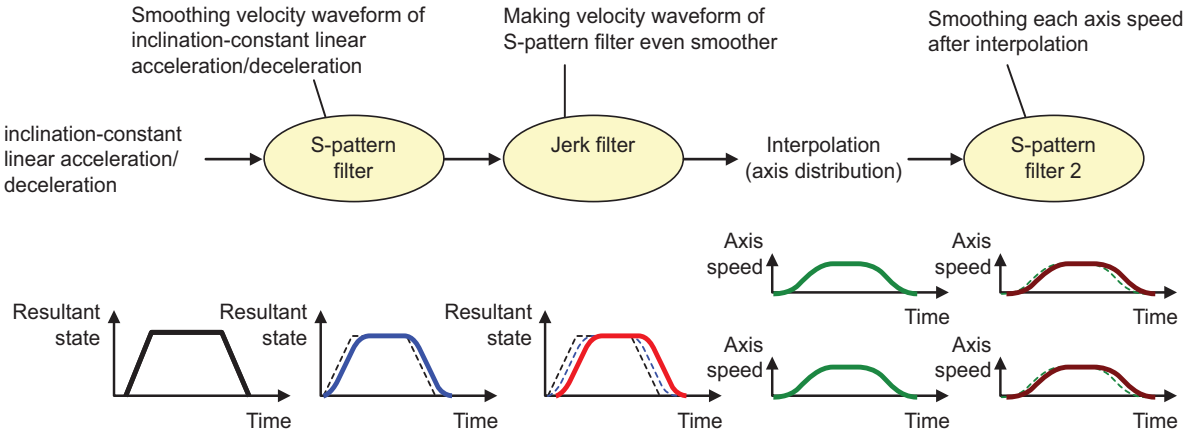
■ NC command and actual tool movement during Feed forward control ON



S-pattern filter control

S-pattern filter (soft acceleration/deceleration filter) is the function that inhibits the machine vibration by smoothing a velocity waveform. There are types of filters as follows:

- G1/G0 S-pattern filter
- Jerk filter
- S-pattern filter 2

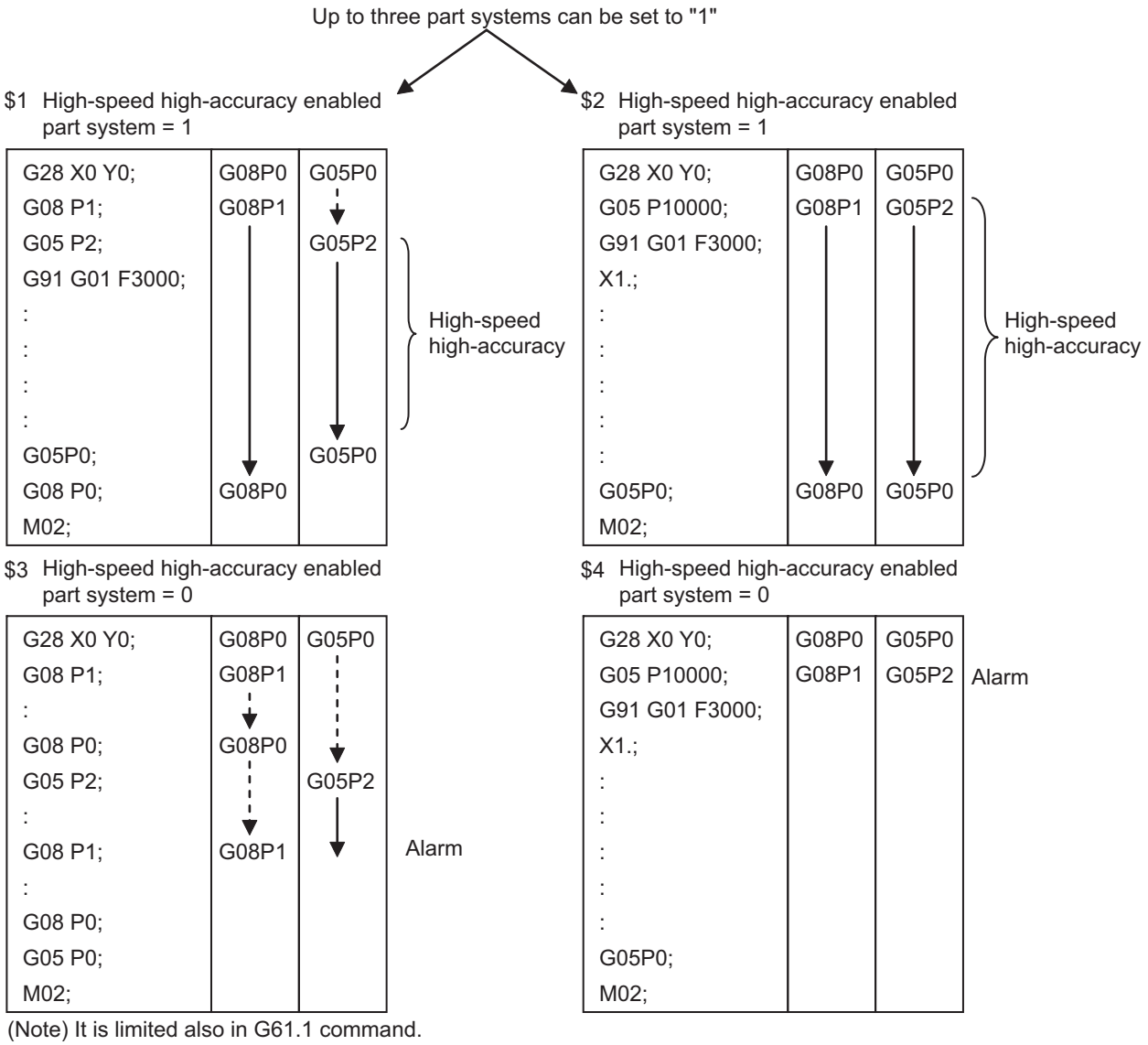


12.3.2.2 Multi-Part System Simultaneous High-Accuracy Control

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ3	Δ3	Δ3	Δ3	○ 3	○ 3	—
L	Δ3	Δ3	Δ3	Δ3	—	—	—

Up to three part systems.

High-accuracy control and high-speed machining mode are available respectively in all part systems, however, the simultaneous usage of high-accuracy control and high-speed machining mode (including High-speed high-accuracy control I/II/III) are available only in part systems which are limited by the parameter "High-speed high-accuracy enabled part system". The high-accuracy control and high-speed machining mode can be used simultaneously in the part system which is set to "1" for "High-speed high-accuracy enabled part system". The program error (P129) occurs when the high-accuracy control and high-speed machining mode are used simultaneously in the part system which is set to "0" for "High-speed high-accuracy enabled part system".
Note that up to three part systems can be set to "1" for "High-speed high-accuracy enabled part system". If four or more part systems are set to "1", MCP alarm (Y51) occurs. If all part systems are set to "0" for "High-speed high-accuracy enabled part system", only the first part system is handled as the one with the parameter set to "1".



(Note) It is limited also in G61.1 command.

12.3.2.3 SSS Control

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

A machining program that approximates a freely curved surface with micro segments can be run at high speed and with high accuracy. This function enables machining with less scratches and streaks on the cutting surface compared to the conventional high-accuracy control function.
When the speed control with reference to the angle between two blocks, such as the conventional high-accuracy control "Optimum corner deceleration", is applied, the acceleration and deceleration may be frequently repeated for local steps and errors, resulting in scratches and streaks on the cutting surface.
The SSS (Super Smooth Surface) control uses the global paths information in addition to the angle information between two blocks. Thus, optimum speed control that is not adversely affected by minute steps or waviness is possible. This reduces the parts which get scratch like marks and streaks on the cutting surface.
Multiple part systems simultaneous high-accuracy function is required to conduct the SSS control in the second or following part systems.

[Features of SSS control]

- This is valid for machining a mold with a smooth shape using a micro segment program.
- This speed control is not easily affected by the error found in the path.
- Even if corner deceleration is not required, the speed is clamped if the predicted acceleration is high.
  (The clamp speed can be adjusted with the parameter "Clamp speed coefficient".)

The length of the path direction recognized with SSS control can be adjusted with the parameter. The range is increased as the setting value increases, and the effect of the error is reduced.

Note

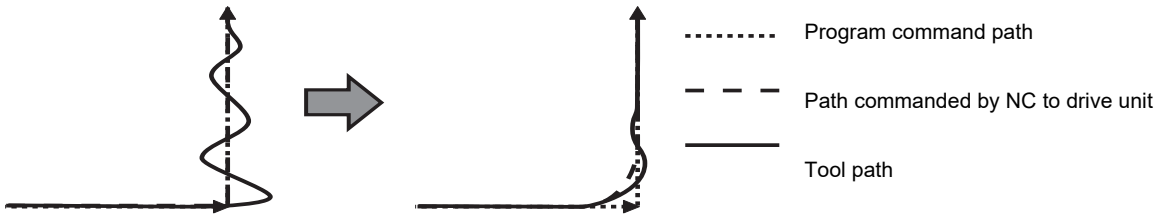
(1) When using this function, one of the following SSS-controlled high-accuracy control functions is required, as well.
- High-accuracy control (G61.1)
- High-accuracy control (G08P1)
- High-speed high-accuracy control I (G05.1Q1)
- High-speed high-accuracy control II (G05P10000)
- High-speed high-accuracy control III (G05P20000)

(2) The performance of micro-segment execution during SSS control differs depending on the type of high-accuracy control mode and machine model to be combined.

12.3.2.4 Tolerance Control

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

This function obtains the optimum clamp speed for corners or curves based on the designated tolerance to perform operations. It also ensures smooth passing in corner sections within the tolerance range, which suppresses machine vibrations. This function allows the machine to operate with the optimum tool path and speed, simply by specifying the tolerance, so an operator can easily carry out high quality machining.
The tolerance refers to the allowable error amount between the path commanded in the machining program and the path output by NC.
This function requires the SSS control specifications because it can only be used under SSS control.



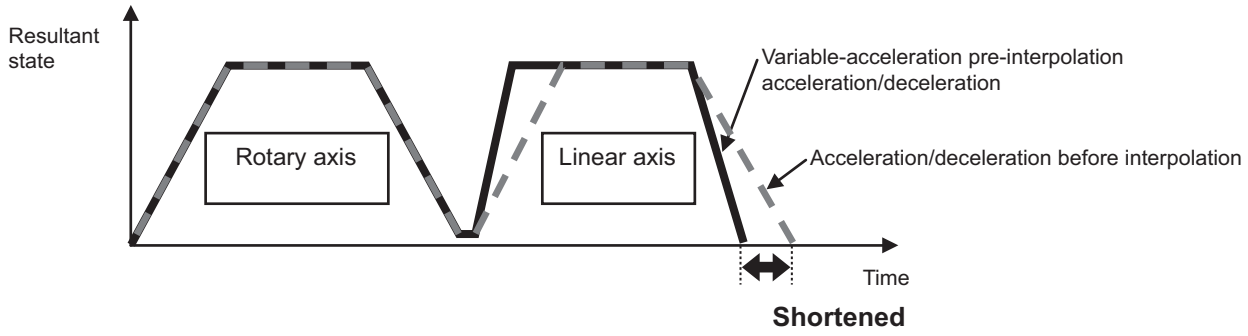
Tolerance control: Invalid　　Tolerance control: Valid

12.3.2.5 Variable-Acceleration Pre-Interpolation Acceleration/Deceleration

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	—	—	—
L	—	—	—	—	—	—	—

This function is useful when each axis differs in the characteristics (responsiveness) during SSS control (4-axis/5-axis machine, etc.).
The normal acceleration/deceleration before interpolation performs the acceleration/deceleration by setting acceleration common to all axes. Therefore, if the high responsiveness and low responsiveness coexist in axes, the acceleration needs to be set to suit the axis with low responsiveness.
On the other hand, the variable-acceleration pre-interpolation acceleration/deceleration can perform the acceleration/deceleration by setting diverse acceleration to each axis. This means that it is possible to set a higher acceleration for axes with high responsiveness than before. Therefore, the cycle time can be reduced especially in the indexing machining.

[Shortening indexing machining time (when responsiveness of linear axis is higher than responsiveness of rotary axis)]



12.3.2.6 High-Accuracy Acceleration/Deceleration Time Constant Extension

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	—	—	—
L	—	—	—	—	—	—	—

The first part system only.

This extends the upper limit of cutting feed time constant from 5,000 [ms] to 30,000 [ms] for acceleration/ deceleration before interpolation.
In the multi-part system, this function cannot be used together as an alarm will occur.

12.3.2.7 Axis-Specific Acceleration Tolerance Control

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	—	—	—
L	Δ	Δ	Δ	Δ	—	—	—

The first part system only.

The acceleration to be generated at a seam between blocks is evaluated for each axis to control deceleration so that the seam is passed at the optimum speed. This enables highly accurate edge machining. The optimum deceleration speed is calculated so that the acceleration of each axis to be generated at the seam is equal to or less than the tolerable acceleration for each axis, which is determined by maximum speed for each axis, time constant for each axis, and the accuracy coefficient. The machine decelerates to the speed in advance, and then accelerates back to the command speed after passing the corner.
This control enables deceleration at an appropriate speed for the characteristics of each axis even when machine vibrations may easily occur due to a low tolerable acceleration for a specific axis (rotary axis). This means that the deceleration speed can be raised at a corner where acceleration is generated only for an axis with a high tolerable acceleration, leading to a reduced cycle time.

12.3.3 High-Speed High-Accuracy Control

A machining program that approximates a freely curved surface with micro segments can be run at high speed and with high accuracy. This function is effective in decreasing the cycle time of machining dies with free curves. This function is also useful in machining which needs to make an edge at a corner or reduce a path error from inner route of curved shape because the high-accuracy control mode is turned ON automatically.

12.3.3.1 High-Speed High-Accuracy Control I (G05.1Q1) Maximum [kBPM]

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ67.5	Δ67.5	Δ67.5	Δ67.5	○ 33.7	○ 33.7	○ 33.7
L	Δ67.5	Δ67.5	Δ67.5	Δ67.5	○ 33.7	○ 33.7	—

■ Command format

G05.1 Q1 ;	High-speed high-accuracy control I ON

G05.1 Q0 ;	High-speed high-accuracy control I OFF

■ Micro segment capacity

The following is the micro segment capacity while high-speed high-accuracy control I is valid.

Micro segment processing capacity (unit: kBPM)

Number of part systems/ Number of axes	Number of high-speed high-accuracy control-enabled part systems	M850V / M830V		M80VW / M80V Type-A		M80V Type-B	
		M	L	M	L	M	L
• 1-part system (8 axes or less)	1 part system	67.5	67.5	33.7	33.7	33.7	—
• 1-part system (9 axes or more) • 2-part system	1 part system	67.5	67.5	33.7	33.7	33.7	—
	2 part systems	33.7	33.7	16.8	16.8	—	—
• 3-part system	1 part system	33.7	33.7	16.8	16.8	—	—
	2 part systems	33.7	33.7	16.8	16.8	—	—
• 4-part system (16 axes or less)	1 part system	33.7	33.7	16.8	16.8	—	—
	2 part systems	33.7	33.7	16.8	16.8	—	—
	3 part systems	16.8	16.8	16.8	16.8	—	—
• 4-part system (17 axes or more) • 5-part system or more	1 part system	16.8	16.8	—	—	—	—
	2 part systems	16.8	16.8	—	—	—	—
	3 part systems	16.8	16.8	—	—	—	—

Note

(1) The above performance applies under the following conditions.
- 3 axes or less commanded simultaneously in G01
- Block containing only axis name and movement amount (Does not contain macro or variable command)
- During tool radius compensation OFF (G40)

It may not be possible to attain the specified feed rate if deviated from the above conditions.

12.3.3.2 High-Speed High-Accuracy Control II (G05P10000) Maximum [kBPM]

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ168	Δ168	Δ168	Δ168	○ 101	○ 101	○ 67.5
L	Δ168	Δ168	Δ168	Δ168	○ 101	○ 101	—

■ Command format

G05 P10000 ;	High-speed high-accuracy control II ON
G05 P0 ;	High-speed high-accuracy control II OFF

■ Micro segment capacity

The following is the micro segment capacity while high-speed high-accuracy control II is valid.

Micro segment processing capacity (unit: kBPM)

Number of part systems/ Number of axes	Number of high-speed high-accuracy control-enabled part systems	M850V / M830V		M80VW / M80V Type-A		M80V Type-B	
		M	L	M	L	M	L
• 1-part system (8 axes or less)	1 part system	168 135 (*3) 101 (*1) 67.5 (*2)(*4)	168 135 (*3) 101 (*1)	101 67.5 (*1) 33.7 (*2)(*4)	101 67.5 (*1)	67.5	—
• 1-part system (9 axes or more) • 2-part system	1 part system	135 101 (*3) 67.5 (*1) 33.7 (*2)(*4)	135 101 (*3) 67.5 (*1)	101 67.5 (*1) 33.7 (*2)(*4)	101 67.5 (*1)	67.5	—
	2 part systems	67.5 33.7 (*2)	67.5	33.7	33.7	—	—
• 3-part system	1 part system	33.7	33.7	33.7	33.7	—	—
	2 part systems	33.7	33.7	33.7	33.7	—	—
• 4-part system (16 axes or less)	1 part system	33.7	33.7	33.7	33.7	—	—
	2 part systems	33.7	33.7	33.7	33.7	—	—
	3 part systems	16.8	16.8	16.8	16.8	—	—
• 4-part system (17 axes or more) • 5-part system or more	1 part system	16.8	16.8	—	—	—	—
	2 part systems	16.8	16.8	—	—	—	—
	3 part systems	16.8	16.8	—	—	—	—

(*1) This value indicates the micro segment processing capacity in the time constant expansion system or in the following modes.
- Tool center point control (G43.4/G43.5)
- Inclined surface machining (G68.2/G68.3)
- Workpiece installation error compensation (G54.4P1 to 7)
- 3-dimensional tool radius compensation (G41.2/G42.2)
- Spatial error compensation
- Variable-acceleration pre-interpolation acceleration/deceleration
- Spline interpolation 2
- 3D machine interference check
- Tool cutting point control (G43.8/G43.9)
- Rotation center error compensation
- Real-time 3D machine interference check

(*2) This value indicates the micro segment processing capacity when Smooth fairing is ON.

(*3) This value indicates the micro segment processing capacity when Tolerance ctrl is ON.

(*4) When Smooth fairing is ON and the control cycle is twice the standard cycle, a program error occurs.

Note

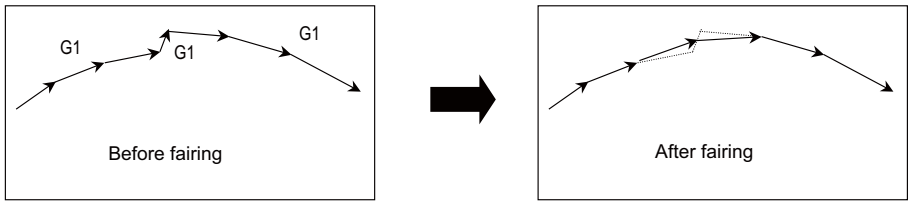
(1) The above performance applies under the following conditions.
- 3 axes or less commanded simultaneously in G01
- Block containing only axis name and movement amount (Does not contain macro or variable command)
- During tool radius compensation OFF (G40)

It may not be possible to attain the specified feed rate if deviated from the above conditions.

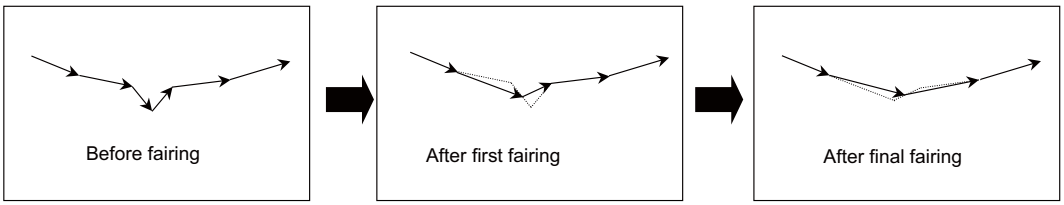
■ Additional functions when high-speed high-accuracy control II/III mode is ON

Fairing

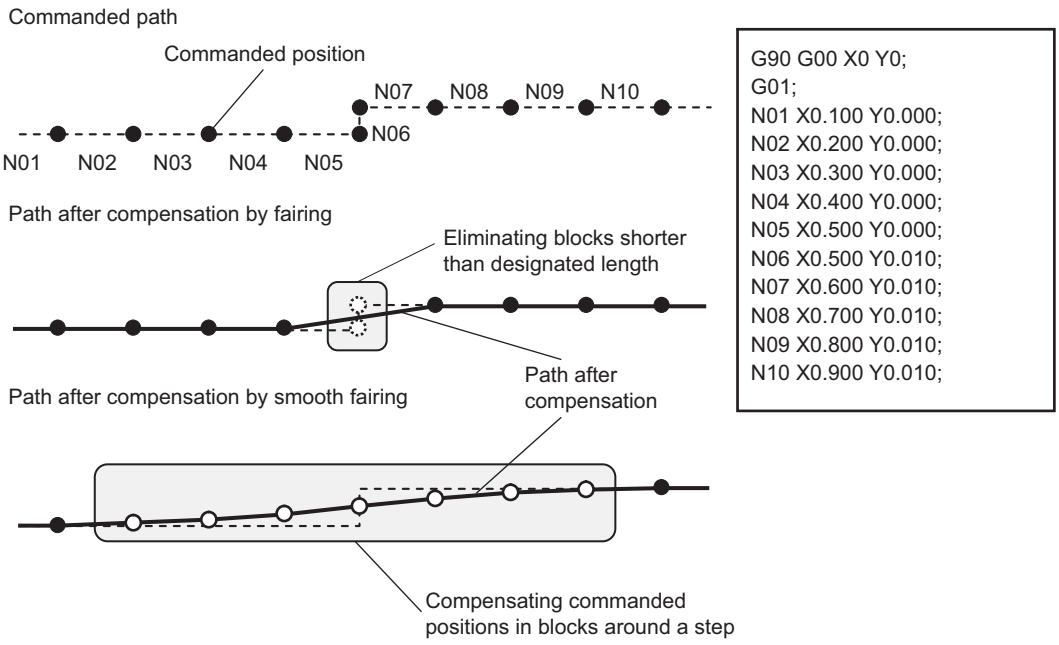
If the protruding path (zigzagging path) is shorter than the parameter setting values in the machining program generated with a CAM, etc., this function can be used to eliminate the protruding paths smaller than the setting value so that the front and back paths are smoothly connected.
This function is valid only for continuous linear commands (G1).



If there is any protruding path after fairing, fairing is repeated.



There is the smooth fairing function in addition to the fairing function.
When a minute step exists on a commanded path, for instance, the path after compensation differs between fairing and smooth fairing as follows:

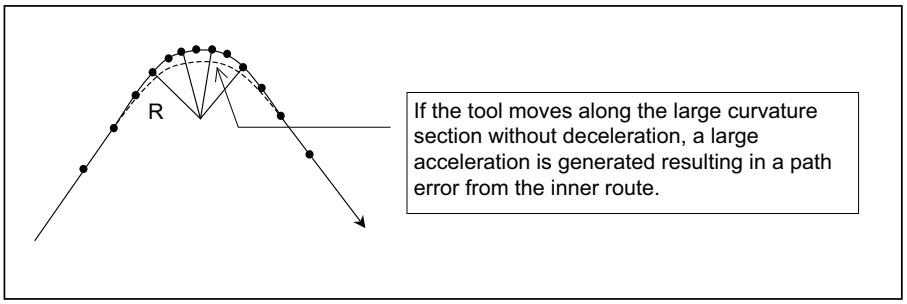


Smooth fairing

This function interpolates the command position of machining program so that the movement path becomes smooth. This function is useful when executing a micro segment program to machine smoothly at low speed or a rough machining program with long segment to machine smoothly.
For details, refer to " 12.3.3.4 Smooth Fairing".

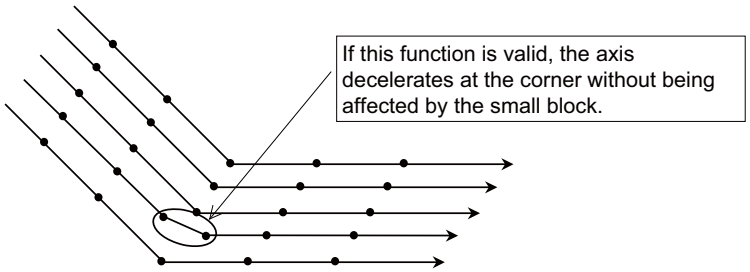
Acceleration clamp speed

With the cutting feed clamp speed during the high-speed high-accuracy control II mode, when the parameter is set, the speed is clamped so that the acceleration generated by each block movement does not exceed the tolerable value. This function clamps the speed optimally even at a section where "angle change at each block is small but entire curvature is large" such as shown below. The tolerable acceleration value is calculated from the parameter setting values.



High-speed mode corner deceleration

Conventionally during high-accuracy control, if the angle is large between the adjacent blocks in the machining program, this function automatically decelerates so that the acceleration generated when passing through the corner is within the tolerable value.
If a small block is inserted at the corner section with the machining program generated with the CAM, etc., the corner passing speed will not match the periphery. This can affect the machining surface.
If this type of small block is inserted when using high-speed mode corner deceleration, the corner will be largely judged by the parameter settings.
The small block is excluded when the angle is judged, but it not excluded from the actual movement command.



12.3.3.3 High-Speed High-Accuracy Control III (G05P20000) Maximum [kBPM]

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ540	Δ540	Δ540	Δ540	○ 202	○ 202	—
L	—	—	—	—	—	—	—

■ Command format

[Setting to enable the function by G code]

High-speed high-accuracy control III can be enabled by the following G code command. If the following G code is commanded, the high-speed high-accuracy control III mode will be ON even the parameter "High speed/accu 3" is OFF.

```
G05  P20000 ;     High-speed high-accuracy control III ON
G05  P0 ;         High-speed high-accuracy control III OFF
```

[Setting to enable the function by parameter]

The high-speed high-accuracy control II command can be treated as the high-speed high-accuracy control III command by enabling the parameter "High speed/accu 3".

```
G05  P10000 ;     High-speed high-accuracy control III ON
G05  P0 ;         High-speed high-accuracy control III OFF
```

G05P2 command in high-accuracy control mode can also be treated as the high-speed high-accuracy control III command.

■ Micro segment capacity

The following is the micro segment capacity while high-speed high-accuracy control III is valid.

Micro segment processing capacity (unit: kBPM)

Number of part systems/ Number of axes	Number of high-speed high-accuracy control-enabled part systems	M850V / M830V		M80VW / M80V Type-A		M80V Type-B	
		M	L	M	L	M	L
• 1-part system (8 axes or less)	1 part system	540 (*1) 404 (*3) 270 101 (*2) 67.5 (*4)(*5)	—	202 67.5 (*2) 33.7 (*4)(*5)	—	—	—
• 1-part system (9 axes or more) • 2-part system	1 part system	168 67.5 (*2) 33.7 (*4)(*5)	—	135 67.5 (*2) 33.7 (*4)(*5)	—	—	—
	2 part systems	101 67.5 (*2) 33.7 (*4)	—	67.5 33.7 (*2)(*4)	—	—	—
• 3-part system	1 part system	33.7	—	33.7	—	—	—
	2 part systems	33.7	—	33.7	—	—	—
• 4-part system (16 axes or less)	1 part system	33.7	—	33.7	—	—	—
	2 part systems	33.7	—	33.7	—	—	—
	3 part systems	16.8	—	16.8	—	—	—
• 4-part system (17 axes or more) • 5-part system or more	1 part system	16.8	—	—	—	—	—
	2 part systems	16.8	—	—	—	—	—
	3 part systems	16.8	—	—	—	—	—

(*1) If the control cycle is twice of the standard cycle, the processing capacity will be 540 kBPM.

(*2) This value indicates the micro segment processing capacity in the time constant expansion system or in the following modes.

- Tool center point control (G43.4/G43.5)
- Inclined surface machining (G68.2/G68.3)
- Workpiece installation error compensation (G54.4P1 to 7)
- 3-dimensional tool radius compensation (G41.2/G42.2)
- Spline interpolation 2
- Tool cutting point control (G43.8/G43.9)

(*3) This is the micro segment processing capacity in the following modes when the control cycle is twice the standard cycle.

- Spatial error compensation
- Variable-acceleration pre-interpolation acceleration/deceleration
- 3D machine interference check
- Rotation center error compensation

(*4) This value indicates the micro segment processing capacity when Smooth fairing is ON.

(*5) When Smooth fairing is ON and the control cycle is twice the standard cycle, a program error occurs.

Note

(1) The above performance applies under the following conditions.

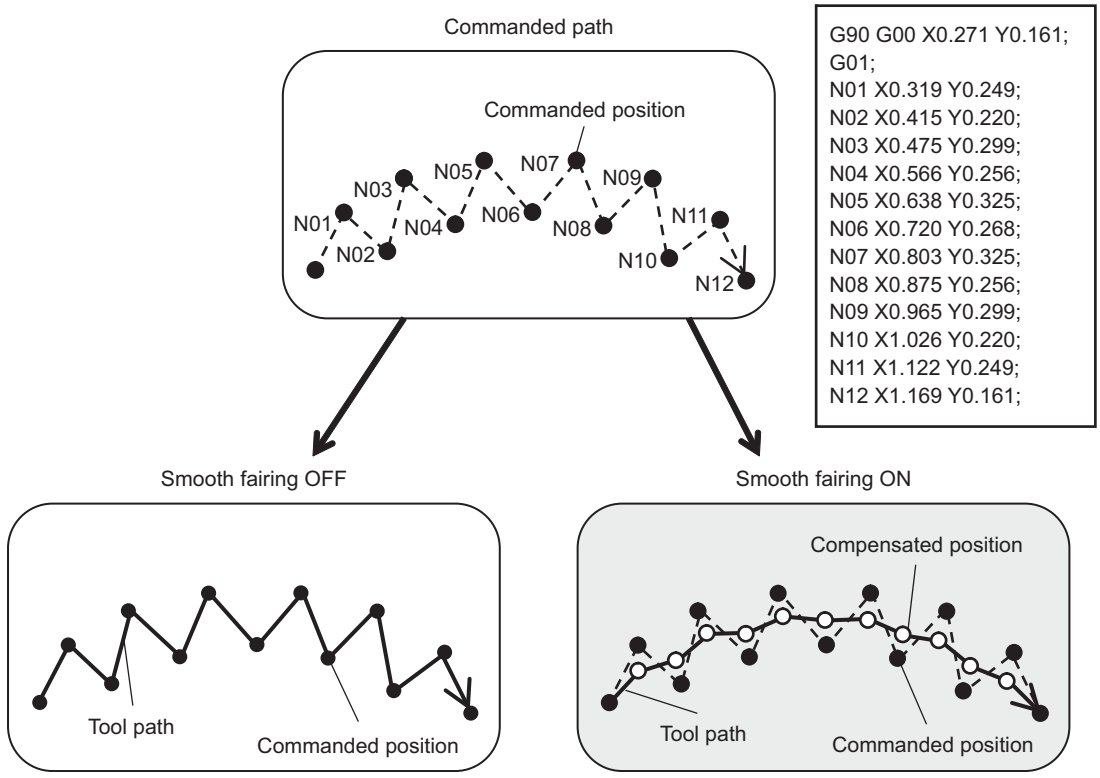
- 3 axes or less commanded simultaneously in G01
- Block containing only axis name and movement amount (Does not contain macro or variable command)
- During tool radius compensation OFF (G40)

It may not be possible to attain the specified feed rate if deviated from the above conditions.

12.3.3.4 Smooth Fairing

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	—
L	—	—	—	—	—	—	—

This function interpolates the command position of machining program so that the movement path becomes smooth. This function is useful when executing a micro segment program to machine smoothly at low speed or a rough machining program with long segment to machine smoothly.
This function is enabled while high-speed high-accuracy control II is ON or while high-accuracy control is ON in high-speed machining mode II/III, and performs compensation in consecutive G01 command during the time.
Refer to " 12.3.3.2 High-Speed High-Accuracy Control II (G05P10000) Maximum [kBPM]" for the differences between fairing and smooth fairing.



Note

(1) While smooth fairing is valid, the high-speed high-accuracy control III operates as high-speed high-accuracy control II.

12.3.4 Machining Condition Selection I

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

The machining condition parameter group which is consists of parameters related to the high-accuracy control can be configured in advance for each machining application (such as part machining or die machining) or machining process (such as rough or finishing), and it can be switched according to the purpose.
This can easily perform the machining which meets the conditions for the intended machining.
The machining condition parameter group can be switched by the following methods.

- Screen setting
- G code
- PLC signal

Multiple part systems simultaneous high-accuracy function is required to apply the machining conditions to the second or following part systems.
The machine condition commanded by G code command will reflect only to commanded part system.

12.3.5 Direct Command Mode

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	Δ	Δ	Δ	Δ	—	—	—

This function reduces the loads of NC machining program analysis and interpolations as much as possible so that machining programs composed of micro segments are executed at high-speed with a processing speed.
Compared to normal machining modes, this function is effective for accelerating 3-dimensional machining of curved surface such as variable cam machining.

12.4 Programming Support Functions

12.4.1 Playback

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	Δ	Δ	Δ	Δ	—	—	—

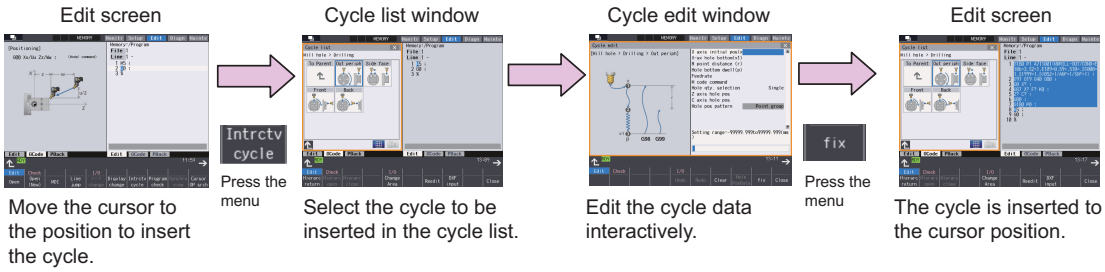
This function enables creation of a program while proceeding with sample machining by manual (handle or job) feed or mechanical handle feed. A machining program can be edited by using the playback movement amount obtained by manual operation as the program command values.
Playback edit is disabled for the machining program in the mass-editing mode.

12.4.3 Interactive Cycle Insertion

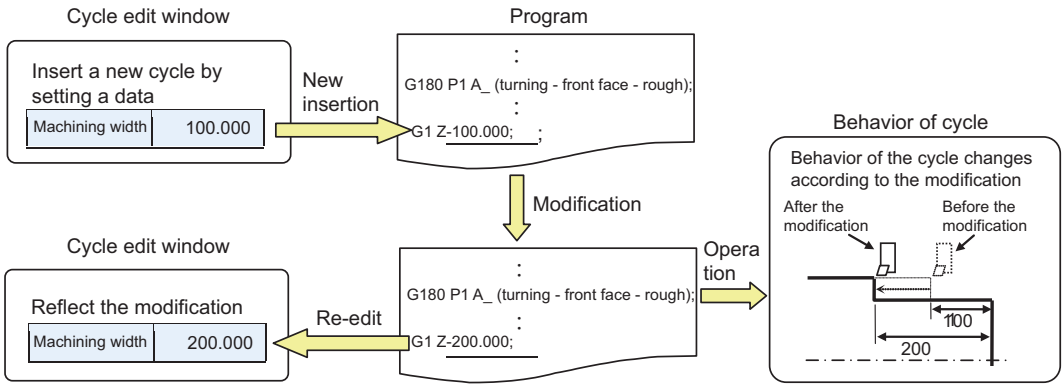
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

This function enables to interactively insert a cycle to assist in the machining and setup for the program opening on the edit screen.
The programming time can be reduced by using this function.
The cycle can easily be inserted by editing data in an interactive window.

[Operation image to insert a cycle]



The program of the cycle block inserted once can directly be edited on the edit screen, and can be re-edited in the cycle edit window if the cycle format is not changed.



[Types of cycle]

Machining cycle can be inserted.
In accordance with the NC system set by the parameter, either cycle for lathe or cycle for machining center can be selected.

For cycle types, there are "Standard type" and "Extend type" of MITSUBISHI cycle and arbitrary cycle type "Custom type" prepared by MTB.
For "Standard type", there are normal format and special format, which is set with the parameter.

Press the menu [Standard type], [Extend type] or [Custom type] to be selected on the interactive cycle screen. Select a cycle type to be changed on the displayed cycle list window to switch the cycle.

[Cycle for Lathe system]

○ : Cycle insertion available - : No specification

Hierarchy 1	Hierarchy 2	Hierarchy 3	Standard type	Extend type
Cycle type	Machining cycle	Machining portion		
Turning hole	Drilling	Front Back	○	○
	Deep-hole drilling		○	○
	Tapping		○	○
	Deep-hole tapping		○	○
	Reaming		—	○
	Boring		○	○
Turning	Turning	Outer diameter Inner diameter Front Back	○	○
	Turning (compound)		○	○
	Copying		○	○
	Grooving		○	○
	Cutting-off		○	○
	Thread cutting	Outer diameter Inner diameter	○	○
	Thread cutting (compound)		○	○
	Thread recutting		○	○
	Thread recutting (compound)		○	○
Milling hole	Drilling	Periphery Side Front Back	○	○
	Deep-hole drilling		○	○
	Step drilling		○	○
	Tapping		○	○
	Deep-hole tapping		○	○
	Reaming		—	○
	Boring		○	○
Milling	Pocketing	Side Front Back	—	○
	Grooving		—	○

Note

(1) For standard type, there are conventional format and Mitsubishi CNC special format. Which format to be used depends on the setting of the parameter.
(2) Fixed cycle is used for "Tapping" and "Deep-hole tapping" of "Extend type". The format to be used is the same as "Standard type", which depends on the setting of the parameter.

[Cycle for Machining center system]

○ : Cycle insertion available - : No specification

Hierarchy 1	Hierarchy 2	Standard type	Extend type
Cycle type	Machining cycle		
Hole	Drilling	○	○
	Deep-hole drilling	○	○
	Step drilling	○	○
	Tapping	○	○
	Deep-hole tapping	○	○
	Reaming	—	○
	Boring	○	○
Milling	Plane	—	○
	Pocketing	—	○
	Grooving	—	○
	Contour	—	○

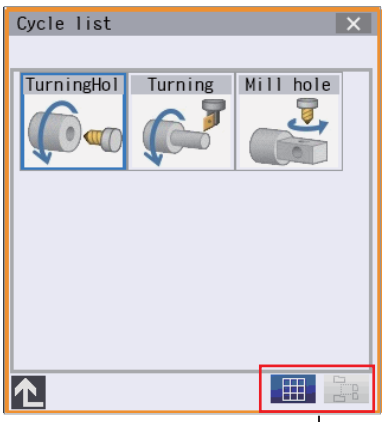
Note

(1) Fixed cycle is used for "Tapping" and "Deep-hole tapping" of "Extend type".

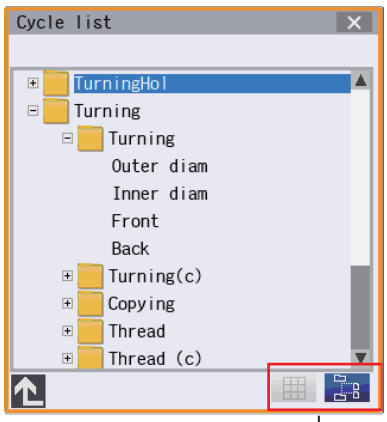
[Display of cycle list window]

The cycle list window is displayed in the area opposite to the editing area. There are two display types for cycle list window, "image display type" and "tree display type", which can be switched Change list button.

Image display type



Tree display type



Change list button

12.4.4 Simple Programming (NAVI MILL/LATHE)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

Create a part program by using NAVI MILL (for machining center system) or NAVI LATHE (for lathe system).

- The following machining processes can be edited.

M system	L system
• Hole drilling (Drilling, deep-hole drilling, step, boring, tapping) • Surface cutting (Circle, square) • Contour cutting (Circle, square, arbitrary shape) • Pocket machining (Circle, square, L pattern, U pattern) • EIA	<Turning machining> • Turning (Outside dia., inside dia., front face) Copying (Outside dia., inside dia., front face) • Threading (Outside dia., inside dia., front face) • Grooving (Outside dia., inside dia., front face) • Trapezoid Grooving (Outside dia., inside dia., front face) • Hole drilling (Drilling, deep-hole drilling, step, tapping) • EIA • Cutting-off <Milling machining> • Milling hole machining (Drilling, deep-hole drilling, boring, tapping) • [Hole pattern] - Random (Front face, outer periphery, side face) - Line (Front face, outer periphery, side face) - Arc (Front face, side face) - Circle (Front face, side face) - Square (Front face, side face) - Lattice (Front face, side face) - Key groove (Front face, outer periphery, side face) • Contour machining (Front face, outer periphery, side face) <Auxiliary operation> • Delivery <Balance cut> • Turning balance cut • Copying balance cut • Two-part system simultaneous thread cutting

- Cutting conditions are automatically determined from tool management data and cutting condition file.
- The operation screen consists of the list view area and the operation view area. In the list view area, the entire part program can be seen at all time. In the operation view area, there are guide drawings related to each input item, which allows an easy data input.
- Program Checker enables the tool paths of a part program to be graphically traced. With this function, an input error can be detected at an earlier stage.
- Guidance function provides an operator with error recovery information.
- Part program is a macro-program-based NC program. Process commands can be added in the edit screen.
- The macro program above can be customized by the machine tool builder.

12.4.5 G Code Guidance

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

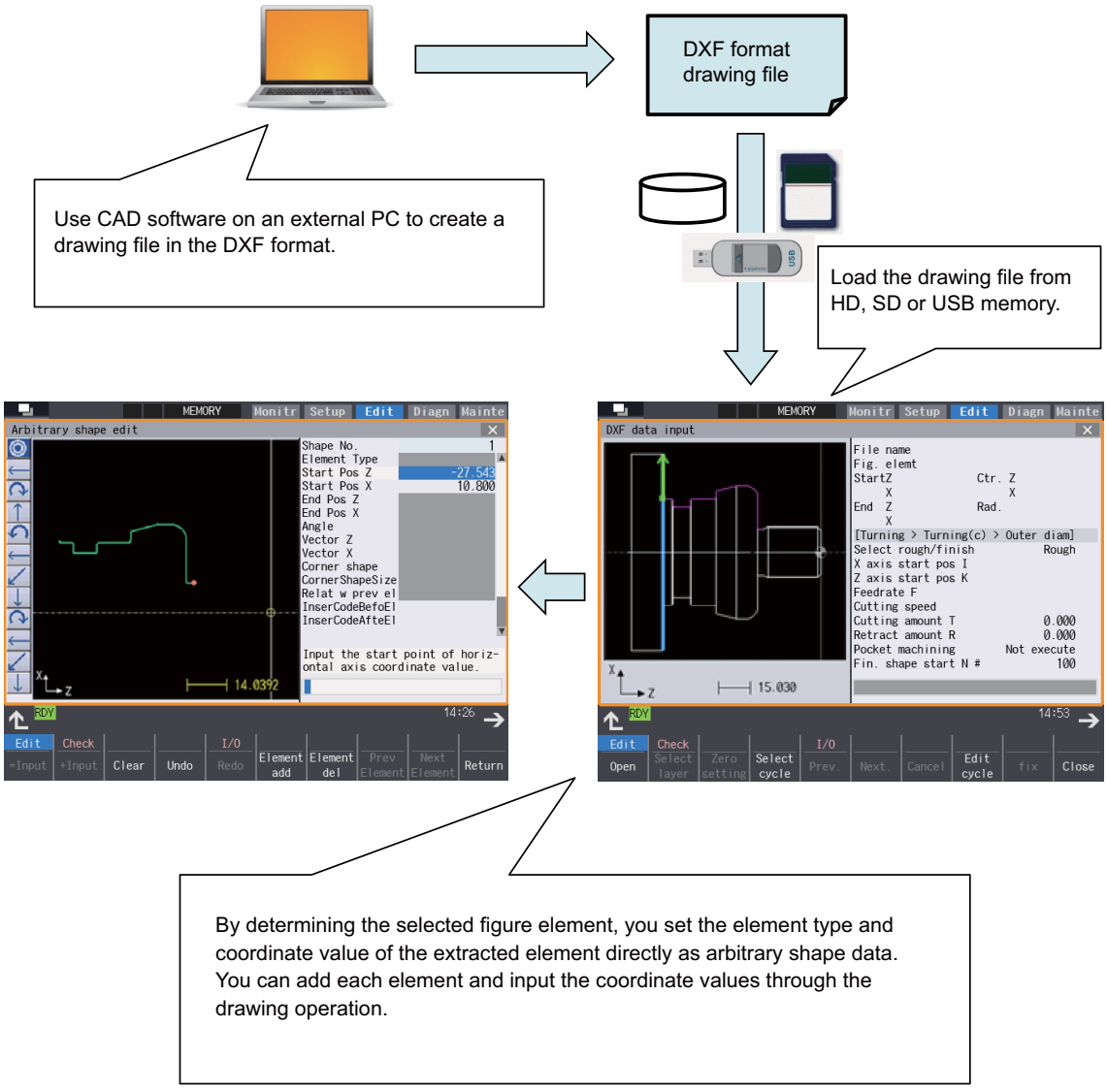
G code guidance is a function to display illustration of the contents or movements of the commanded format for the G code currently under editing. This is used when creating or editing a machining program.
With this function, the G code contents under the editing process can be checked on the spot.

12.4.7 DXF Data Input

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	—
L	Δ	Δ	Δ	Δ	○	○	—

This function is a support for the interactive cycle insert function.
This function allows you to import a DXF drawing file from an external I/O device through DXF data input window, and the figure element data is extracted from the drawing. You can add them as arbitrary shape data or as hole position data to the cycle being edited using the interactive cycle insertion function.

The supported devices are as follows:
- HD
- Memory card (front-side SD card)
- DS (back-side SD card)
- USB memory



12.4.8 Process Analysis

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

This 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.

13

Machine Accuracy Compensation

13.1 Static Accuracy Compensation

13.1.1 Backlash Compensation

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

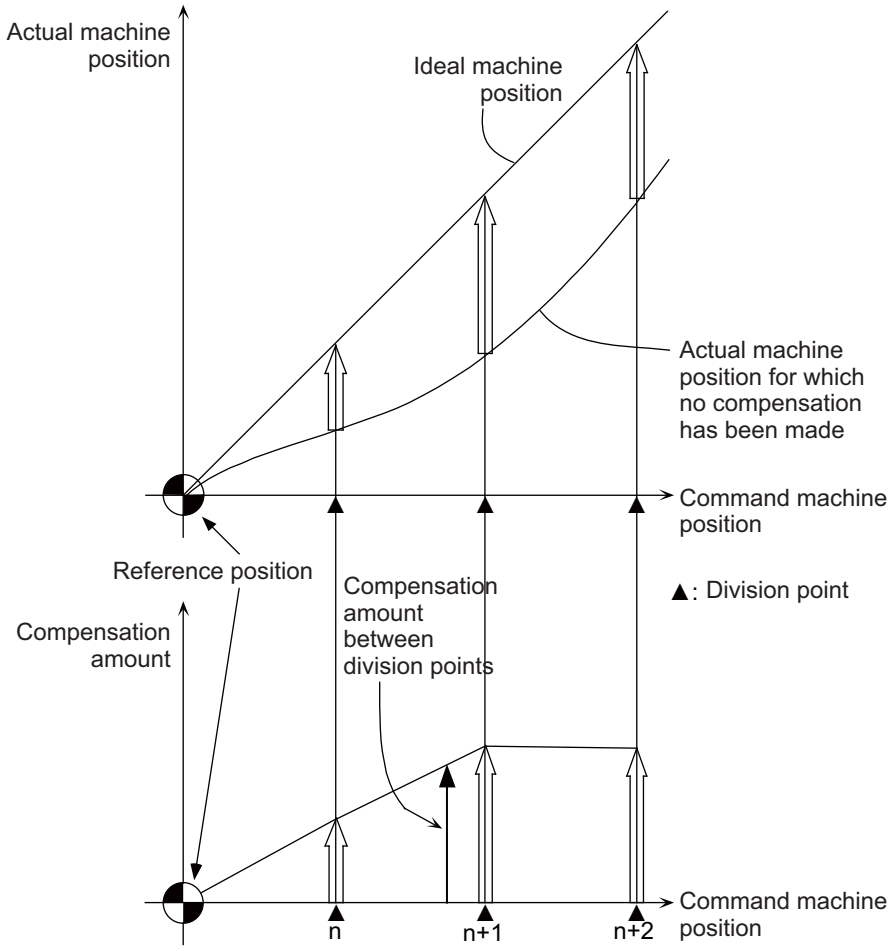
This function compensates for the error (backlash) produced when the direction of the machine system is reversed.
The backlash compensation can be set in the cutting feed mode or rapid traverse mode.
The amount of backlash compensation can be set separately for each axis. It is set using a number of pulses in increments of one-half of the least command increments. The output follows the output unit system. The "output unit system" is the unit system of the machine system (ball screw unit system).

13.1.2 Memory-Type Pitch Error Compensation [Sets]

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○ 32	○ 32	○ 32	○ 32	○ 16	○ 16	○ 16
L	○ 32	○ 32	○ 32	○ 32	○ 16	○ 16	○ 16

With the reference position defined as the base, set the compensation amount in the division points obtained by equally dividing the machine coordinates.
According to the specified parameters, this method compensates an axis feed error caused by a ball screw pitch error, etc.
The compensation amount can be set by either the absolute or incremental system. Select the desired method with the parameter.
Between the division point n and the division point n + 1, the compensation amount of both division points is smoothly compensated by linear approximation.
Up to 32 sets of axes can be set for the compensation for M800V Series, or up to 16 sets of axes for M80V Series.
Up to 1899 points can be set for the compensation.
PLC axes and PLC indexing axes can be also compensated.

[Relationship between the compensation amount and machine position]



Note

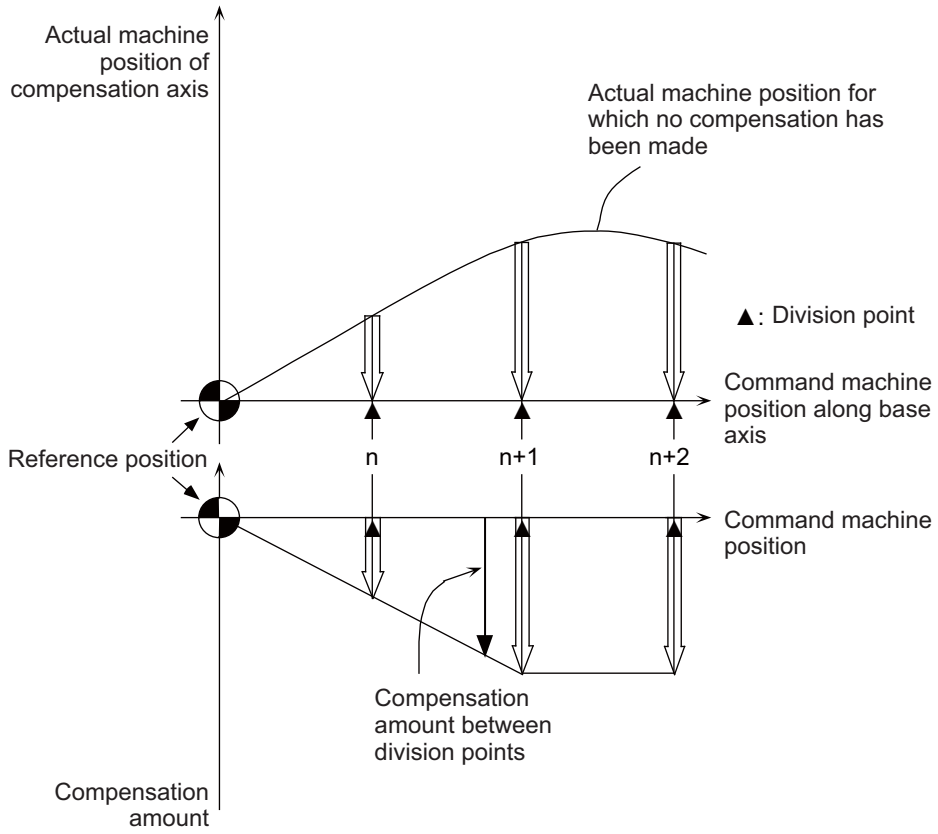
(1) Compensation points 1899 is a total including the points for memory-type relative position error compensation.
(2) A scale of 0 to 99-fold is applied on the compensation amount.

13.1.3 Memory-Type Relative Position Error Compensation

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

This method, according to the parameters specified in advance, compensates the relative position error between two orthogonal axes caused by deflection of the moving stand.
Specify the compensation amount in the compensation axis direction in the division points obtained by equally dividing the machine coordinates of the base axis.
The base axis is one of the two orthogonal axes to which relative position compensation applies. This axis is used as the criterion for relative-error measurement. The compensation axis is the coordinate axis that is orthogonal to the base axis. The compensation is actually made for this coordinate axis.
Machine position between division points n and n+1 is smoothly compensated by the linear approximation of the compensation amount.
Up to 32 sets of axes can be set for the compensation for M800V Series, or up to 16 sets of axes for M80V Series.
Up to 1899 points can be set for the compensation.
PLC axes and PLC indexing axes can be also compensated.

[Relationship between the compensation amount and machine position]



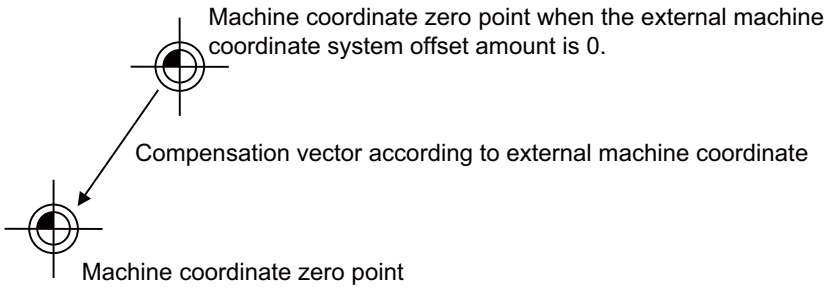
Note

(1) Compensation points 1899 is a total including the points for memory-type pitch error compensation.
(2) A scale of 0 to 99-fold is applied on the compensation amount.

13.1.4 External Machine Coordinate System Compensation

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

The coordinate system can be shifted by inputting a compensation amount from the PLC. This compensation amount will not appear on the counter (all counters including machine position). If the machine's displacement value caused by heat is input for example, this can be used for thermal displacement compensation.



13.1.5 Circular Radius Error Compensation

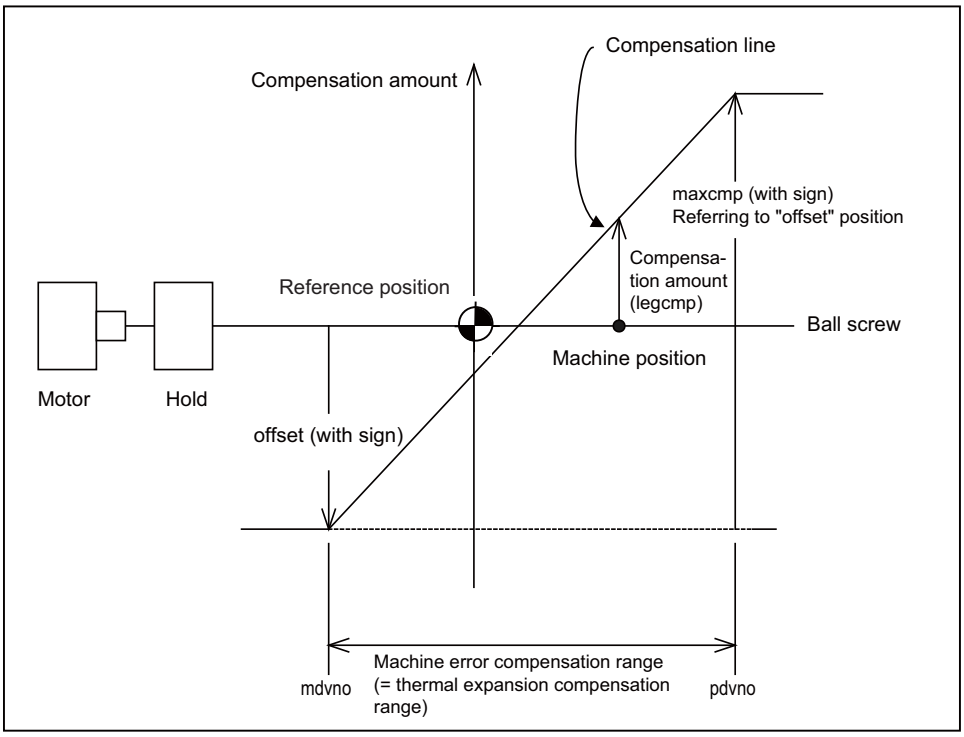
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

With commands designated during arc cutting, this function compensates for movement toward the inside of the arcs caused by a factor such as servo delay.

13.1.6 Ball Screw Thermal Expansion Compensation

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

This compensates the axis feed error caused by the ball screw thermal expansion, etc. using the values set in the R register's thermal displacement compensation parameter.
The compensation amount at the end of the machine error compensation range (mdvno to pdvno) is set. The compensation line is obtained from the set compensation amount, and is linearly distributed in the machine error compensation range to attain smooth compensation. A compensation amount that changes during operation can also be compensated.



■ Conditions under which thermal displacement compensation is valid

- When function is valid.
- When axis No. and parameter (offset, maxcmp) is set in R register.
- When pitch error compensation parameter is set correctly.

■ Compensation operation

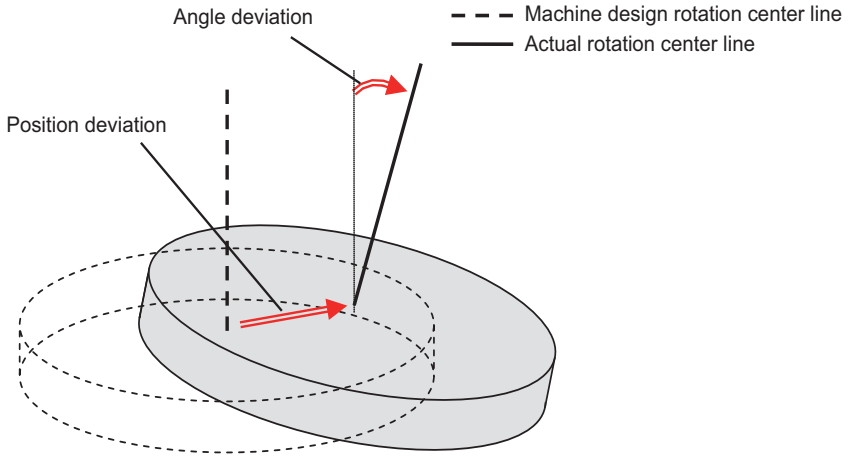
- The machine error compensation range is compensated with the thermal displacement compensation amount (absolute position compensation amount) obtained with the compensation line.
- The compensation amount is immediately calculated when thermal displacement compensation is validated.
- When the compensation amount is changed, the new compensation line is immediately obtained, and the range is compensated again with that compensation amount.
- Thermal displacement compensation can be used with machine error compensation. In this case, the sum of the machine error compensation amount and the thermal displacement compensation amount is added to the machine position and issued as the NC command.

13.1.7 Rotation Center Error Compensation

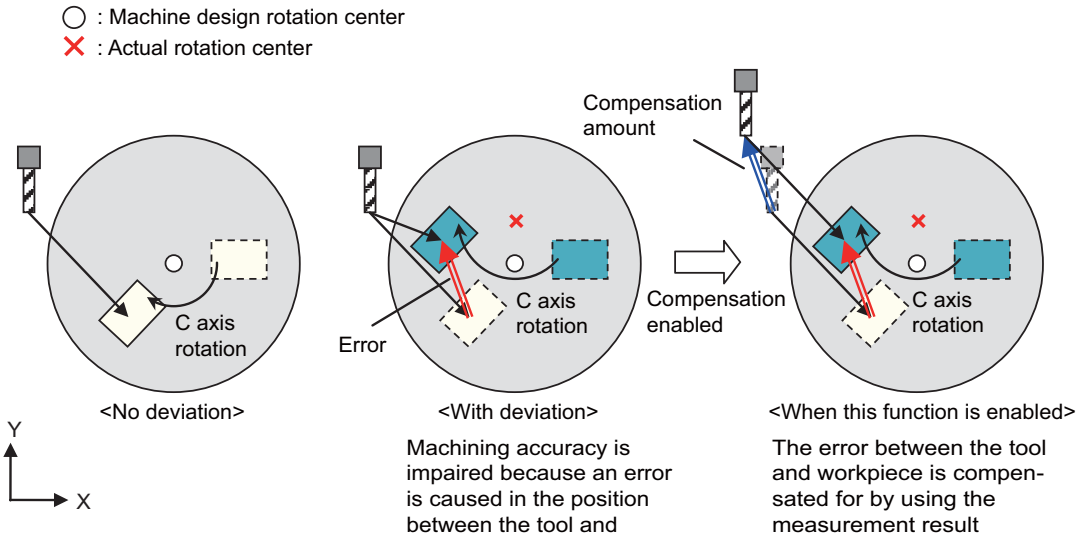
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	—
L	Δ	Δ	Δ	Δ	—	—	—

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.
The deviation can be calculated and set to the parameter by using the rotation center error measurement function.

[Position deviation and angle deviation]



[Compensation for the deviation of the rotation center]



<No deviation>

<With deviation>

Machining accuracy is impaired because an error is caused in the position between the tool and workpiece.

<When this function is enabled>

The error between the tool and workpiece is compensated for by using the measurement result (parameter) of the deviation.

This function can be used on the machine whose rotary axis is installed with ISO polarity and the machine whose axis is installed with non-ISO polarity. However, the orthogonal axis needs to be installed with ISO polarity.(*1)

(*1) In the machine coordinate system (right-hand orthogonal coordinate system) defined in ISO standard, the rotation turning to the right-handed screw is handled as ISO polarity and the one turning to the left-handed screw is handled as non-ISO polarity.

This function is compatible with the following axis configurations of the machines.

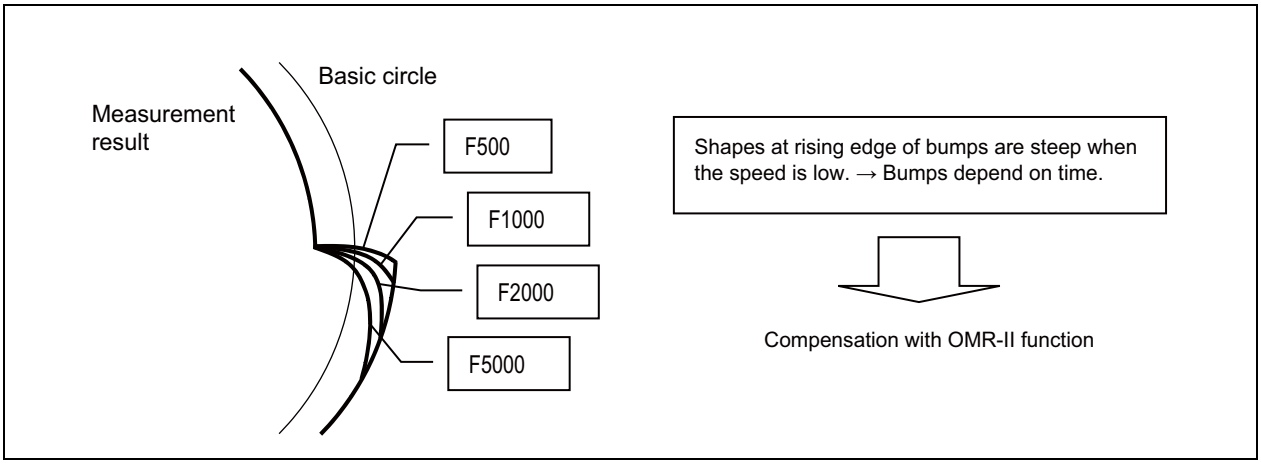
Machine		Machine type	Axis configuration
M system	4-axis machine	Table tilt type	Configured in right-handed orthogonal coordinate system + one rotary axis
		Tool tilt type	
	5-axis machine	Table tilt type	Configured in right-handed orthogonal coordinate system + two rotary axes
		Combined type	
		Tool tilt type	
L system	4-axis machine	Table tilt type	Configured in right-handed orthogonal coordinate system + one rotary axis
		Tool tilt type	
	5-axis machine	Combined type	Configured in right-handed orthogonal coordinate system + two rotary axes
		Tool tilt type	

13.1.8 Position-Dependent Gradually Increasing-Type Backlash Compensation

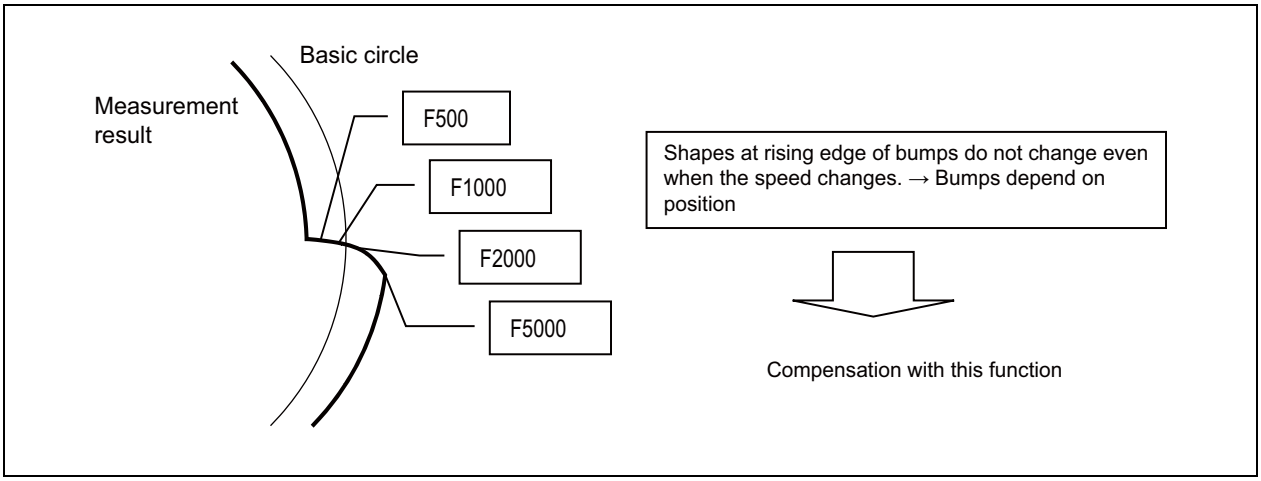
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

Usually, minute bumps are generated when the machine movement direction is reversed. (Lost motion) However, due to various mechanical system factors, there may be cases where the bumps do not change in steps but gradually change after the direction reversal. (Gradually increasing-type lost motion)
Depending on the mechanical structure, the gradually increasing-type lost motion includes the type where the bump amount depends on "duration of time" from when the machine movement direction is reversed and the type where the bump amount depends on "distance". When the bump amount depends on "duration of time", compensation is possible with the OMR-II function. (" 13.2.4 OMR II (Backlash with Filter)").
With this function, the gradually increasing-type lost motion which depends on the distance from the point where the machine movement direction is reversed can be compensated by controlling the variation of backlash compensation amount according to the distance from the direction reversal point.

[When the bump amount changes according to speed]



[When the bump amount does not change according to speed]



13.1.9 Bidirectional Pitch Error Compensation

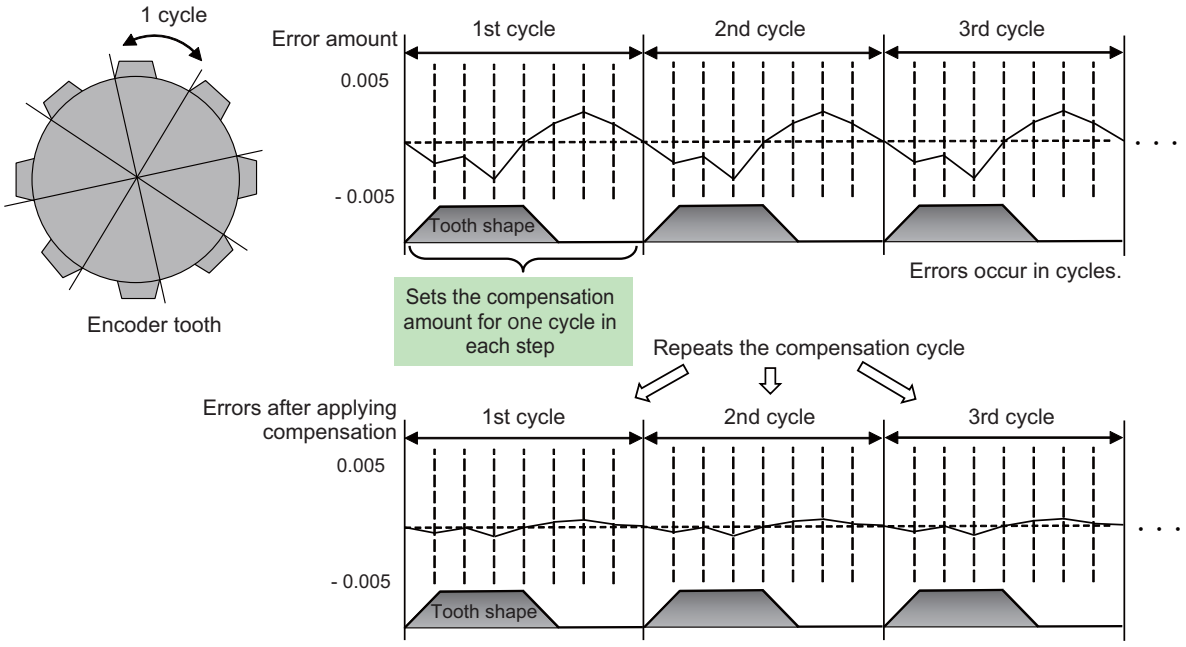
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

Bidirectional pitch error compensation function is used to compensate the pitch error in each direction by setting the pitch error compensation amount when moving in the positive and negative direction. This function reduces the difference of tool path between the positive and negative direction.

13.1.10 Cyclic Error Compensation

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

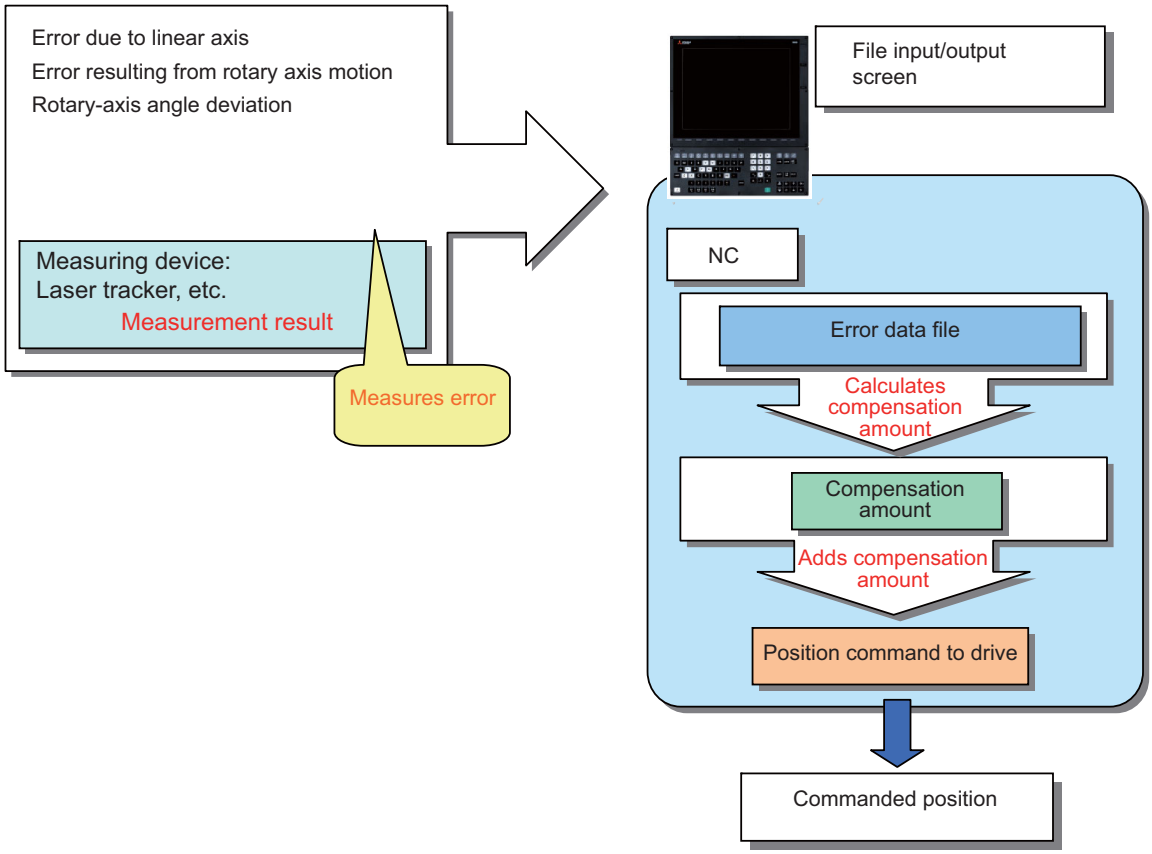
When the machine operation induces certain cyclic errors, this function compensates a series of cyclic errors by setting the compensation amount for a single cycle.
Short-pitched compensation is possible for the errors occurring due to the gear shape of the encoder.
You can set the period of the error cycle, the number of steps in the cycle (maximum 128 points), and the compensation amount at each step by the parameter.



13.1.11 Spatial Error Compensation

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	—
L	—	—	—	—	—	—	—

This function can compensate for three-dimensional errors of a machine tool due to its linear and rotary axes. This measures the spatial errors caused by the linear and rotary axes with a measuring device, and inputs the measurement results to the NC to make an error data file. The function calculates the compensation amount based on the error data file and adds the calculated compensation amount of the linear and rotary axes to the drive command position to perform the compensation. The function can reduce deterioration of machining accuracy due to mechanical errors, which will lead to high-accuracy machining.
This function is enabled for a machine where three linear axes form a right-handed orthogonal coordinate system.



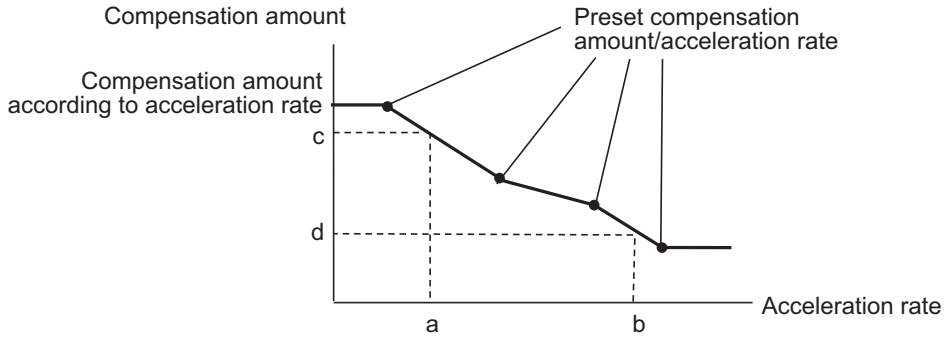
This function is compatible with the following axis configurations of the machines.

Machine	Machine type	Axis configuration
3-axis machine	3 orthogonal axes	Configured in right-handed orthogonal coordinate system
4-axis machine	Table tilt type	Configured in right-handed orthogonal coordinate system + one rotary axis
	Tool tilt type	
5-axis machine	Table tilt type	Configured in right-handed orthogonal coordinate system + two rotary axes
	Combined type	
	Tool tilt type	

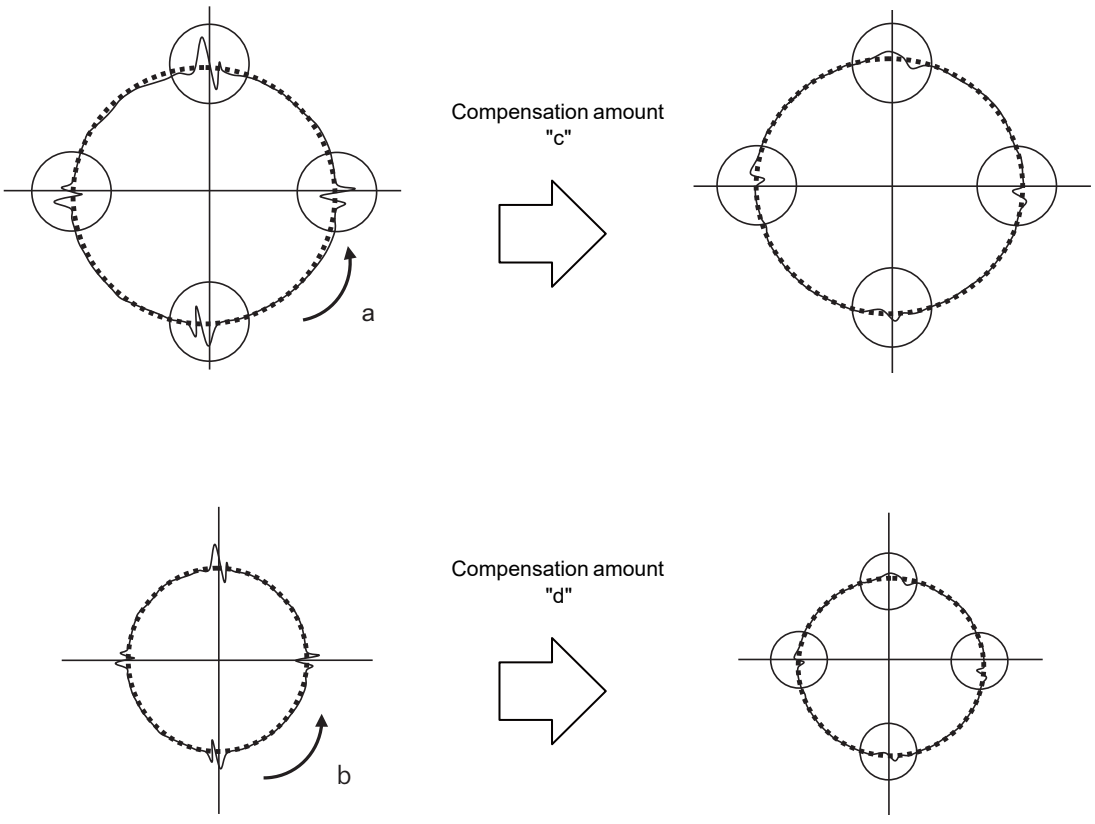
13.1.12 Backlash Compensation II

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	—	—	—	—	—	—	—

This function reduces quadrant error caused by the backlash when the traversing direction is reversed in machines where semi-closed loop control is implemented.
Compensation based on the acceleration rate at a direction reversal improves machining accuracy. For the range between the acceleration rates defined in parameters, the error is compensated using the compensation amount obtained through linear interpolation.
Backlash compensation II allows up to four compensation amounts for different acceleration rates to be set for each axis.



[Before the compensation] **[After the compensation]**



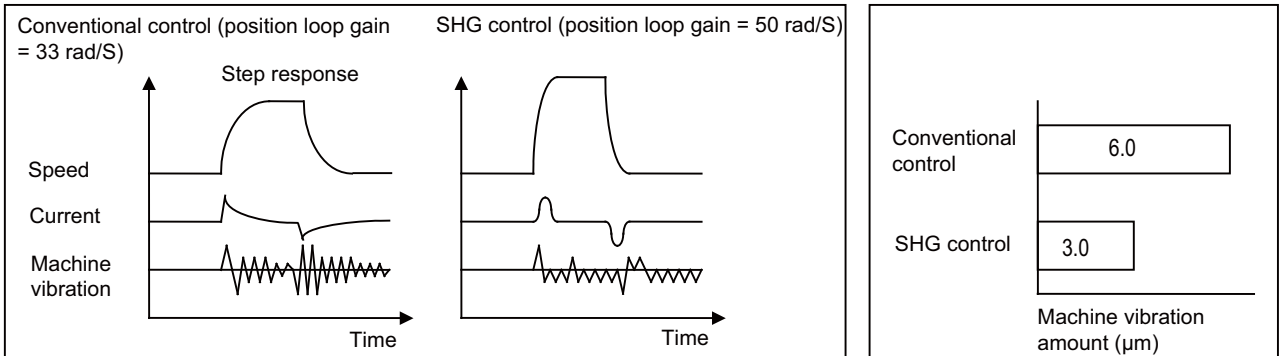
13.2 Dynamic Accuracy Compensation

13.2.1 Smooth High-Gain (SHG) Control

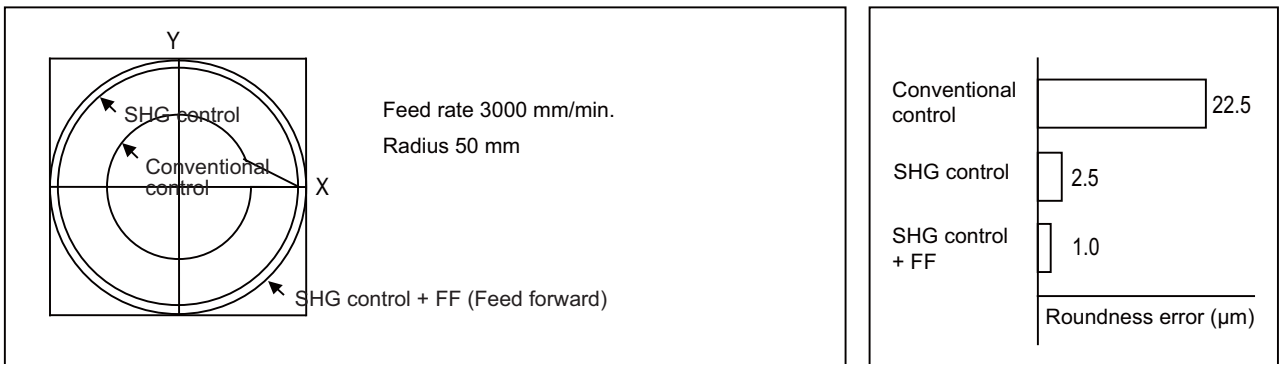
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

This is a high-response and stable position control method using the servo system. This SHG control achieves an approximately three-fold position loop gain equally compared to the conventional control method.
The features of the SHG control are as follows.

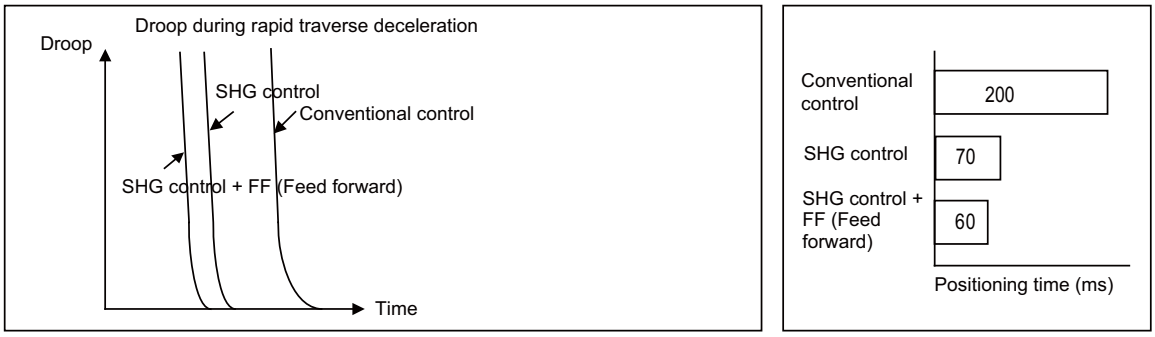
- The acceleration/deceleration becomes smoother, and the mechanical vibration can be suppressed (approximately 1/2) during acceleration/deceleration.
  (In other words, the acceleration/deceleration time constant can be shortened.)



- The shape error is approximately 1/9 of the conventional control.



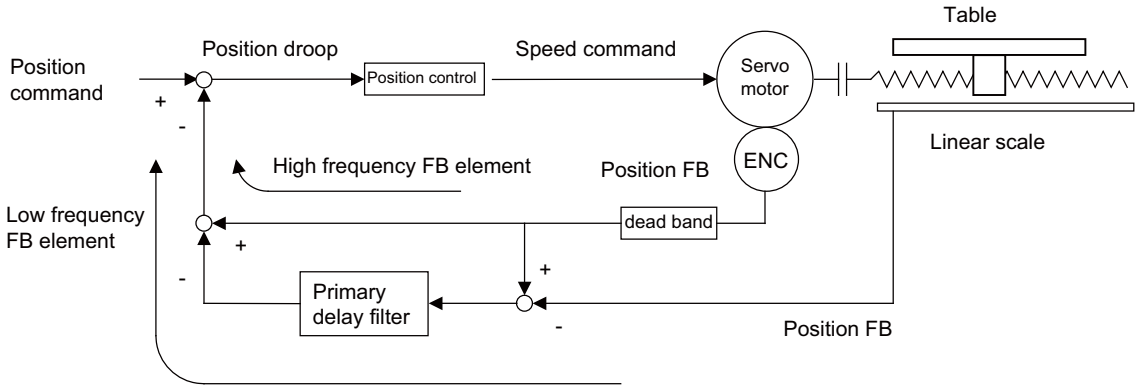
- The positioning time is approximately 1/3 of the conventional control.



13.2.2 Dual Feedback

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

If the motor and machine coupling or machine system's rigidity is low when a closed loop system such as a large machine is used, the position loop gain may not increase due to overshoot caused by the vibrational response during acceleration/ deceleration. The dual feedback function is effective in this case.
To validate the dual feedback function, use position feedback with a motor side detector in ranges with high acceleration to enable stable control. In ranges with low acceleration, use position feedback with the machine side detector (scale). This will make it possible to increase the position loop gain.
The machine side detector (scale) is required separately.



Dual feedback control

The state will approach the semi-closed loop system as the primary delay filter's time constant increases, so the position loop gain limit will increase. Note that the limit of the position loop gain increased with the dual feedback function is the same as the position loop gain limit for a semi-closed system that does not use a machine side detector (scale, etc.). In addition, the positioning time will increase as the primary delay filter time constant increases.

13.2.3 Lost Motion Compensation

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

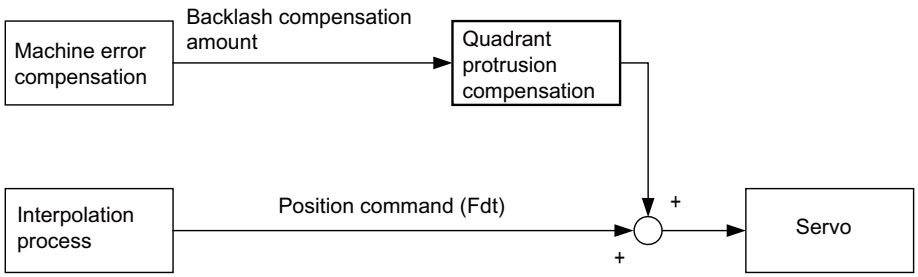
This function compensates the error in the protrusion shape caused by lost motion at the arc quadrant changeover section during circular cutting.

13.2.4 OMR II (Backlash with Filter)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

The OMR (Optimal Machine Response) control function estimates the machine or motor model (moment of inertia, clone friction, viscosity coefficient, etc.) that can cause a path error (error in actual tool path in respect to the path commanded with the program). High-accuracy machining is achieved by carrying out feed forward control based on that model. This allows error cased by quadrant protrusions during circular interpolation or quadrants on the inner side of the path to be greatly reduced.
OMR-II is a function that focuses on the quadrant protrusions, and improves the path error with this. Quadrant path compensation is included in OMR-II.

The quadrant protrusion compensation function improves quadrant protrusions by issuing compensation to the backlash compensation amount to compensate the error when the machine system direction is reversed.
By adding the compensated backlash compensation amount to the position command and sending it to the servo, the gradually increasing-type lost motion which occurs at quadrant changeover can be compensated.



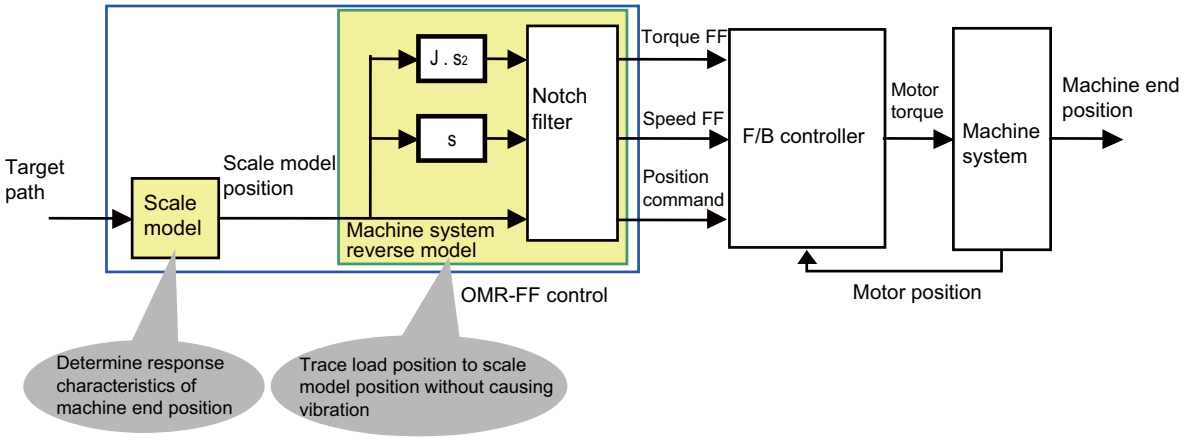
13.2.6 OMR-FF

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

The OMR (Optimal Machine Response) control estimates the causes (moment of inertia, clone friction, viscosity coefficient, etc.) of the path error (error in actual tool path in respect to the path commanded with the program) by making a model of the control target. Feed forward control is carried out based on that model.

With the OMR-FF control method, highly accurate machining can be achieved by using the command filter function to suppress vibration in addition to the reverse model feed forward function.
The reverse model feed forward function estimates the control target's vibration characteristics using a reverse model. The feed forward command for the appropriate position, speed and current can be obtained from this estimation. This allows the machine position to be tracked to the commanded position without causing vibration when using machine tools with low rigidity.
When the command filter function is also used, the high-range vibration elements in the command can also be suppressed.

New control method to bring out the full potential of a high-speed high-acceleration machine
- High tracking ability for the command (Smaller path error)
- Suppression of the machine vibration (Suppress low-range vibration without losing tracking ability)



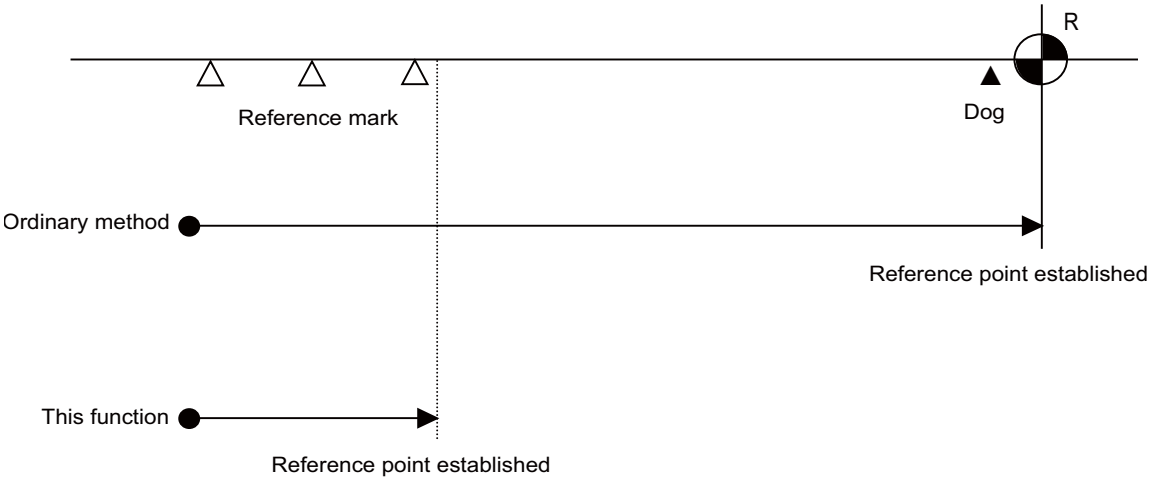
[Setting parameters]
- Scale model time constant
- Machine system resonance frequency ωp, anti-resonance frequency ωz, attenuation rate ζ, inertia J

13.2.7 Distance-Coded Reference Position Detection

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

This is the function where the scale with absolute address reference mark is used to establish the reference point in the relative position detection system.
When the reference point has not been established, it is necessary to move the axis to the reference point. On the other hand, with this function, the reference point is established by moving the axis only for several reference marks, resulting in a significant reduction of the axis movement amount.
No dog is used as the position is calculated using reference marks.

[Reference point establishment process]



13.2.8 OMR-CC (Optimum Machine Response - Contour Control)

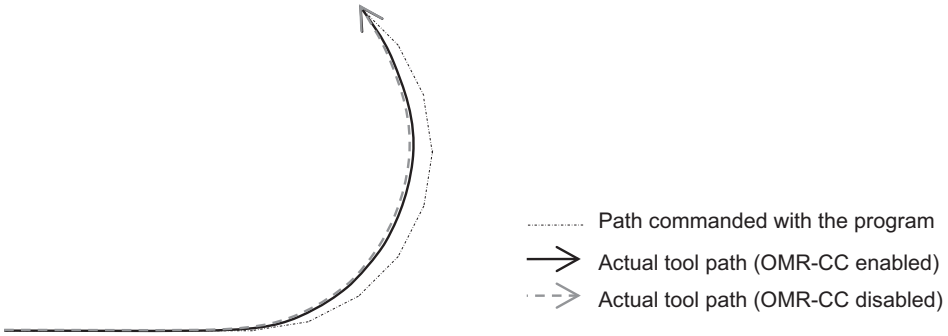
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	△	△	△	△	○	○	○
L	△	△	△	△	○	○	○

OMR-CC (optimum machine response - contour control) is a function that improves the passing speed in curved parts such as arcs and free curved surfaces without increasing the path error (difference between the program path and the tool path) caused by the delay of the servo system. By enabling this function, the machining time can be shortened while maintaining the machining accuracy.

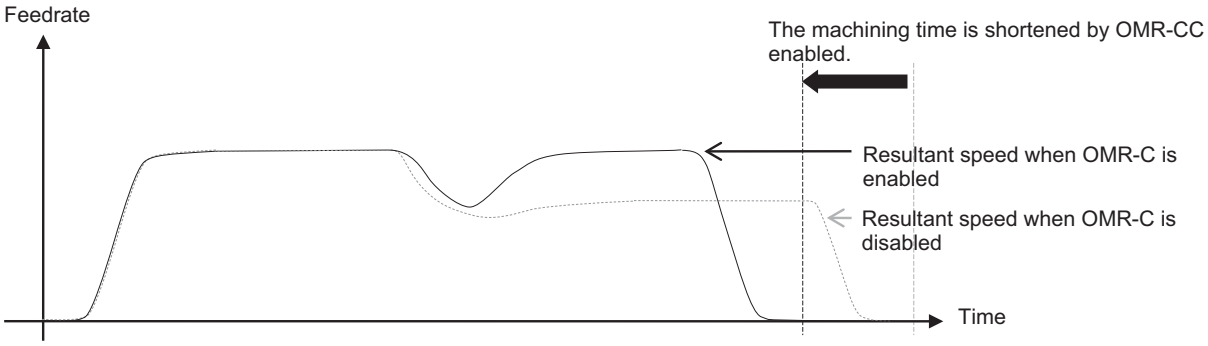
The path and feedrate when OMR-CC is enabled and when OMR-CC is disabled (OMR-FF is enabled) are outlined below.

[Operation example (Path and speed waveform of an arc-shaped machining)]

- Path to pass the curved parts when OMR-CC is enabled/disabled



- Resultant speed to pass the curved parts when OMR-CC is enabled/disabled

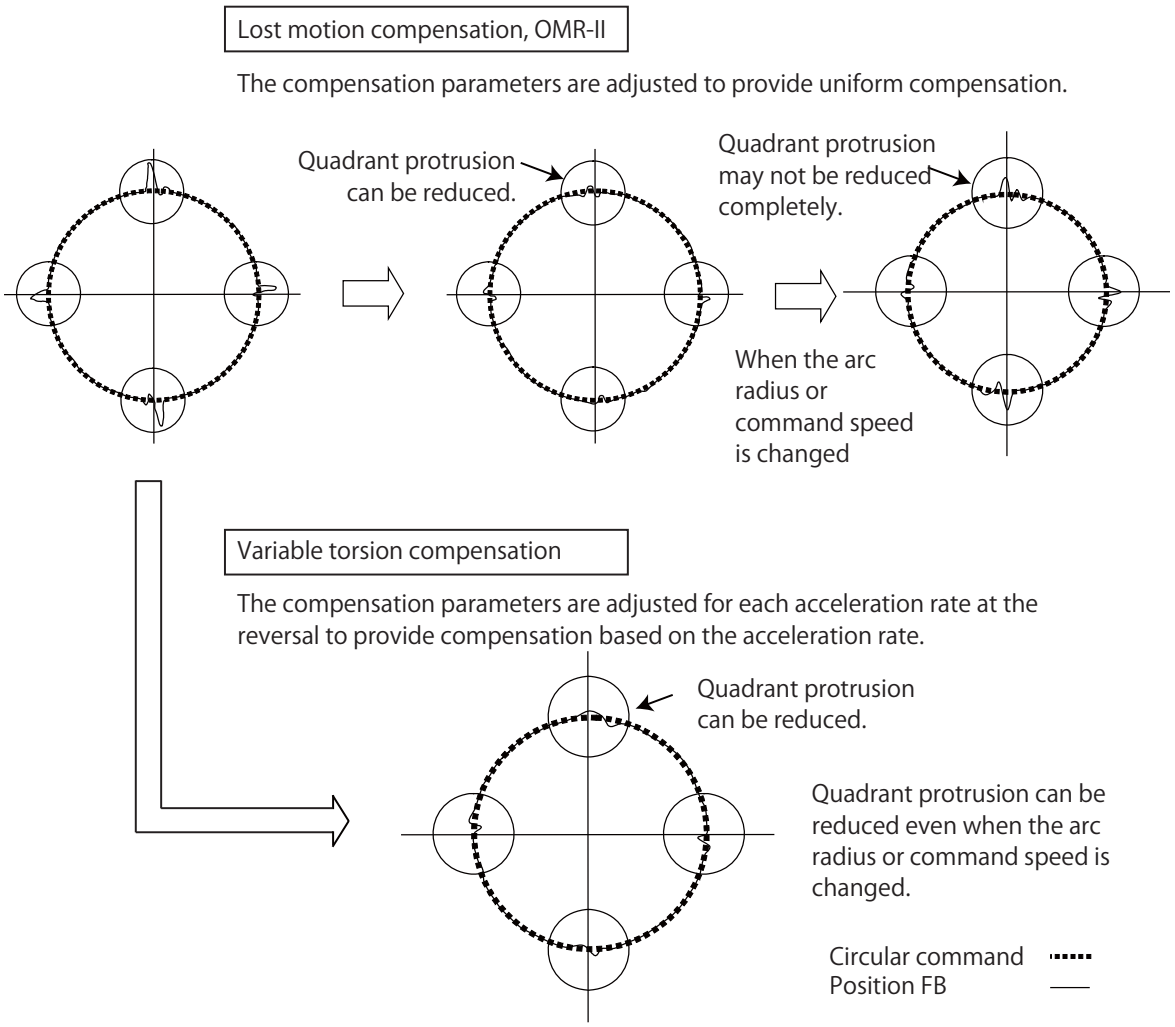


13.2.9 Variable Torsion Compensation

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	—
L	—	—	—	—	—	—	—

When the direction of the machine movement is reversed, a response delay (*1) causes protrusion on the path.
(*1) The response delay occurs due to dead band, such as friction, torsion, and backlash.

The lost motion compensation and OMR-II provide uniform compensation using the set compensation parameters regardless of the acceleration rate at the direction reversal to suppress the protrusion. However, the characteristics of the response delay at the reversal differs depending on the acceleration rate at the reversal; therefore, when the arc radius or command speed is changed, the compensation cannot be performed sufficiently.
The variable torsion compensation enables compensation based on the acceleration rate at the reversal and machining with high accuracy by setting the compensation parameters for each acceleration rate at the reversal.



13.2.10 Cogging Torque Compensation

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

This function compensates the torque ripple (cogging torque).

Ensure to adjust this function before adjusting the machine.
As cogging torque varies with the characteristics of the motors used, machines need to be adjusted individually.

This function is designed for servo motors. The function cannot be used for spindle motors.
This function is valid for the axes in the rotary axis spindle mode during spindle-mode rotary axis control.
This function is invalid for the axes under spindle-mode servo motor control.

14

Automation Support Functions

14.1 Measurement

14.1.1 Skip

14.1.1.1 Skip

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

When the external skip signal is input during linear interpolation with the G31 command, the machine feed is stopped immediately. Then, the coordinate value is read, the remaining distance is discarded and the commands in the next block are executed.
Multiple skip operation is performed with a logical product of multiple signals.

■ Command format

[Skip]

```
G31 X__ Y__ Z__ α__ R__ F__ ;
```

X, Y, Z, α	Axis coordinate value Use absolute or incremental command according to the G90/G91 modal. α is the additional axis.
R	Acceleration/deceleration command R0: Acceleration/deceleration time constant = 0 (Automatic acceleration/deceleration is not performed after interpolation.) R1: Acceleration/deceleration time constant is enabled. Acceleration/deceleration is performed at the time constant set with the parameters. When it is omitted, R0 is commanded (Acceleration/deceleration time constant = 0).
F	Feedrate (mm/min)

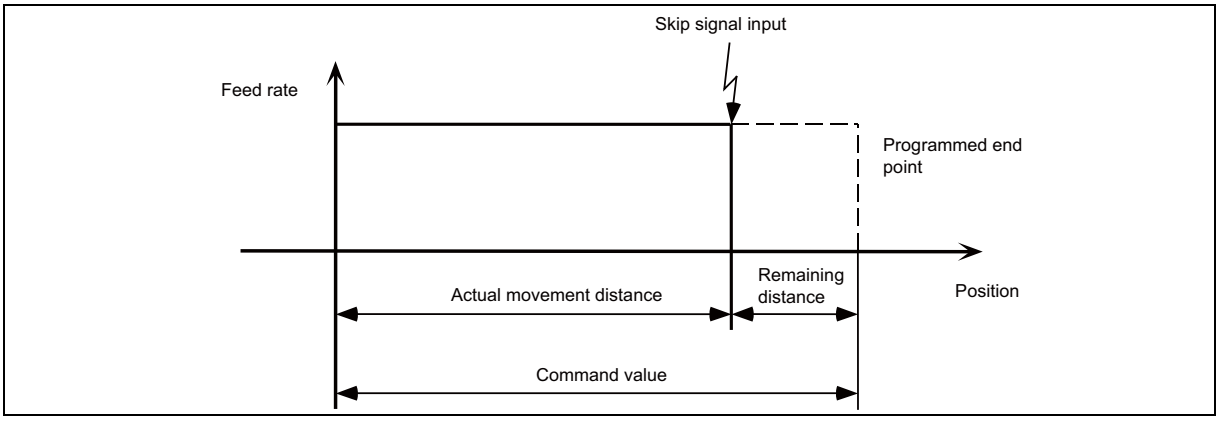
With this command, linear interpolation is performed. When the external skip signal is input during this command, the machine is stopped, the remaining commands are canceled, and the commands in the next block are executed.

[Multiple skip]

G31 D1 X__ Y__ Z__ α__ P__ R__ F__ ;

X, Y, Z, α	Axis coordinate value
D	Skip signal command method D1: Multiple skip (skip operation with logical product (AND) of "P__") When D0 is commanded or D command is omitted, multiple-step skip 2 (G31 Pn method) is performed.
P	Skip signal command
R	Acceleration/deceleration command
F	Feedrate (mm/min)

The skip signals that can be used are 1 to 8. Which signal of these to be used is designated with skip signal command P. When D1 is commanded, if all the signals in the combination specified with skip signal command P are input, the skip operation is performed.



14.1.1.2 Multiple-Step Skip

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

This function realizes skipping by designating a combination of skip signals for each skip command.

■ G31.n method

This function carries out skipping by designating a combination of skip signals for each skip command (G31.1, G31.2, G31.3). The combination of the skip signals are designated with parameters for each G code (G31.1, 31.2, 31.3), and the skip operation is executed when all signals in the combination are input.

G31.n Xx1 Yy1 Zz1 Ff1 ;

G31.n	: Skip command (n=1, 2, 3)
Xx1,Yy1,Zz1	: Axis address and target position
Ff1	: Feedrate (mm/min)

■ G31Pn method

As with the G31.n method, the valid skip signal is designated and skip is executed. However, the method of designating the valid skip signal differs.
The skip signals that can be used are 1 to 8. Which is to be used is designated with P in the program. Refer to Table 1 for the relation of the P values and valid signals.
Skip can be executed on dwell, allowing the remaining dwell time to be canceled during the dwell command (G04) and the next block executed under the skip conditions (to distinguish external skip signals 1 to 8) set with the parameters.

G31 Xx1 Yy1 Zz1 Pp Ff1 ;

G31	: Skip command
Xx1,Yy1,Zz1	: Axis address and target position
Pp	: Skip signal command
Ff1	: Feedrate (mm/min)

- Specify the skip rate in command feedrate F. However, F modal is not updated.
- Specify skip signal command in skip signal command P. Specify the P value in the range of 1 to 255. If it exceeds the specified range, a program error occurs.
- When the skip signals are commanded in combination, the skip operation takes place with OR result of those signals.

Table 1 Valid skip signals

Skip signal command P	Valid skip signal							
	8	7	6	5	4	3	2	1
1								○
2							○	
3							○	○
4						○		
5						○		○
:	:	:	:	:	:	:	:	:
253	○	○	○	○	○	○		○
254	○	○	○	○	○	○	○	
255	○	○	○	○	○	○	○	○

14.1.1.4 PLC Skip

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

This function enables skip operations to be performed by signals which are input from the user PLC.

14.1.1.5 Speed Change Skip

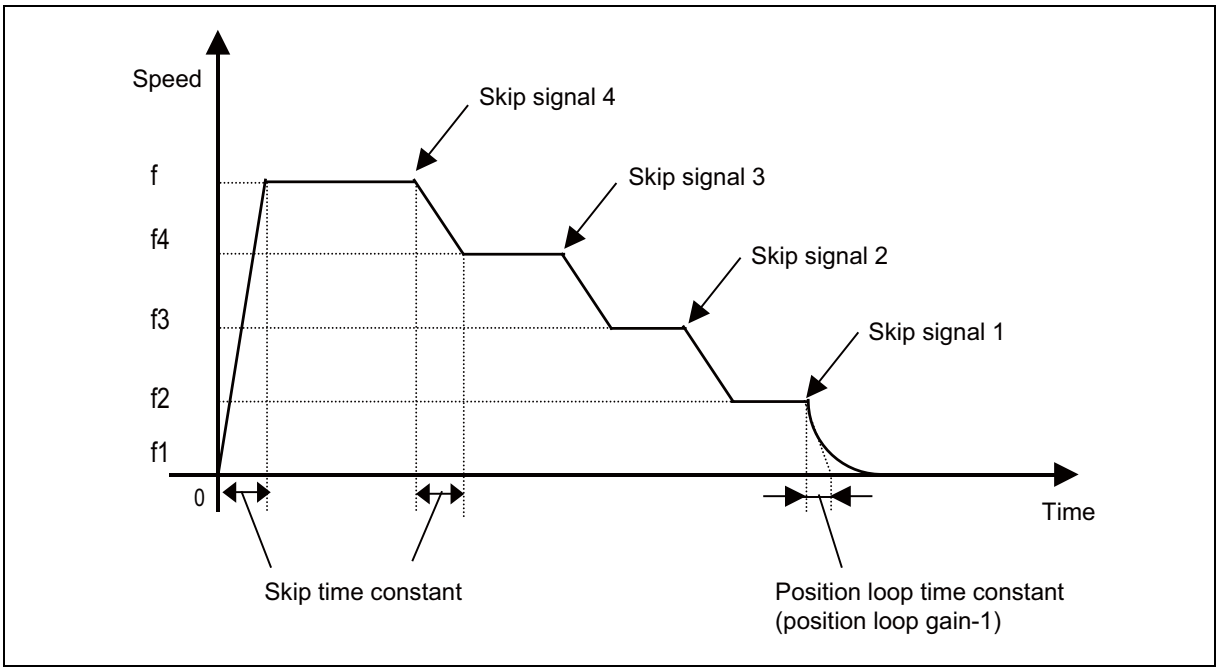
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	—	—	—
L	Δ	Δ	Δ	Δ	—	—	—

This function is used to change the feed rate or to stop the movement by inputting the skip signal during the linear interpolation.
After stopping the movement with skipping, the remaining movement command is canceled and the next block is executed.
In addition to the speed change skip function, the high-speed skip option and multiple-step skip function are required to use this function.

G31 Xx Yy Zz Ff F1 = f1 ... Fn = fn ; ("n" indicates skip signals 1 to 8.)

G31	Skip command
Xx, Yy, Zz	Command position
Ff	Feedrate when starting the cutting feed (mm/min)
fn	Feedrate after detecting the skip signal (mm/min) fn = 0 : Movement stop fn ≠ 0 : Changing the feed rate to fn F1 = Feed rate after inputting the skip signal 1 : F8 = Feedrate after inputting the skip signal 8

Operation example
G31 X100. Ff F1 = 0 F2 = f2 F3 = f3 F4 = f4 ;

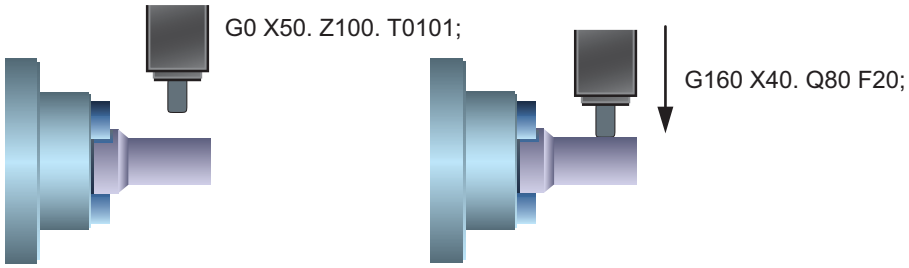


14.1.1.6 Torque Limitation Skip

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

Axis movement is performed in the torque limited status, and the axis movement command is suspended to proceed to the next block when the current command value reaches the designated torque skip value and the torque skip turns ON.
In addition to the torque, the droop value can be add to the condition of the skip ON. (Droop skip)
This function enables measurement without a sensor.

Workpiece radius measurement tool




```
:
G0 X50. Z100. T0101;   Tool selection for measurement
G160 X40. Q80 F20;     Torque skip command
#100=#5061; :          Coordinate position (workpiece value) read
:
```

■ Command format

G160 X/U/Z/W/α Q D F ;	
X/U/Z/W/α	: Axis address (range of coordinate position command (mm, inch), decimal point command is possible)
Q	: Torque skip value (0 to 500 (%))
D	: Droop skip value (0 to 99999.999 (mm, inch))
F	: Skip speed (range of feedrate (mm/min, inch/min, mm/rev, inch/rev))

14.1.2 Automatic Tool Length Measurement

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

This function moves the tool in the direction of the tool measurement position by assigning a distance between the measurement start position to the measurement position. The machine stops when the tool reaches the sensor and automatically calculates the difference between the coordinate position where the tool stopped and measurement position. It registers this difference as the tool length compensation amount for that tool.

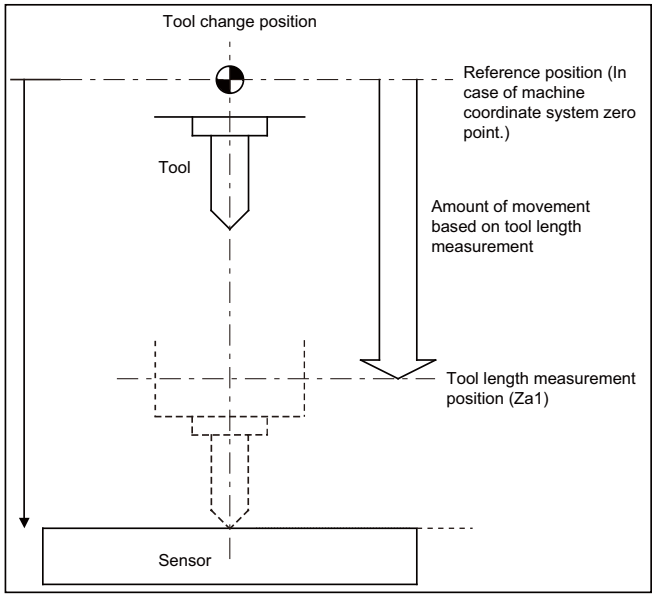
Automatic tool length measurement (M system)

This function moves the tool in the direction of the tool measurement position by commanding a distance between the measurement start position to the measurement position, it stops the tool as soon as it contacts the sensor and calculates the difference between the coordinate position when the tool has stopped and commanded coordinate position. It registers this difference as the tool length compensation amount for that tool.
If compensation has already been applied to the tool, it is moved in the direction of the measurement position with the compensation still applied, and when the measurement and calculation results are such that a further compensation amount is to be provided, the current compensation amount is further corrected.
If the compensation amount at this time is one type, the compensation amount is automatically corrected; if there is a distinction between the tool length compensation amount and wear compensation amount, the wear amount is automatically corrected.

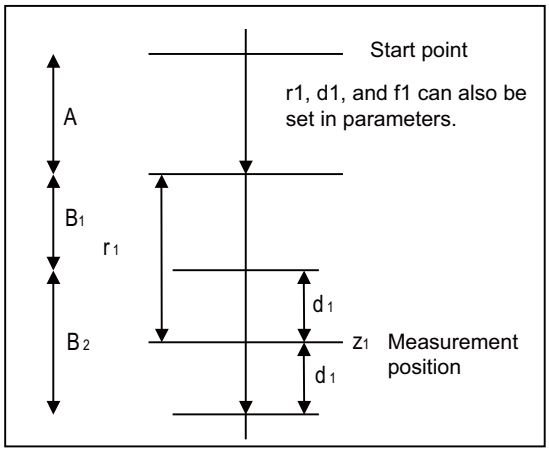
G37 Zz1 Rr1 Dd1 Ff1 ;

G37 ;	: Measurement command
Zz1	: Measurement axis address and measurement position X, Y, Z, α (α = optional axis)
Rr1	: The distance between the point at which tool movement is to start at the measurement speed and the measurement position
Dd1	: The range in which the tool is to stop
Ff1	: The measurement rate

When Rr1_, Dd1_ and Ff1_ have been omitted, the values set in the parameters are used.



At this time, the tool length compensation amount has a minus ("-") value.
Program example
G28 Z0 ;
T01 ;
M06 T02 ;
G43 G00 Z0 H01 ;
G37 Z-300. R10. D2. F10 ;
:
In this case, the distance (H01 = Za1 - z0) from the tool T01 tip to the top of the measurement sensor is calculated as the tool length compensation amount which is then registered in the tool compensation table.



Area A: Moves with rapid traverse feed rate.
Areas B_1 and B_2: Moves with the measurement speed. (f1 or parameter setting)
If a sensor signal is input in area B_1, an error will occur.
If a sensor signal is not input in the area B_2, an error will occur.

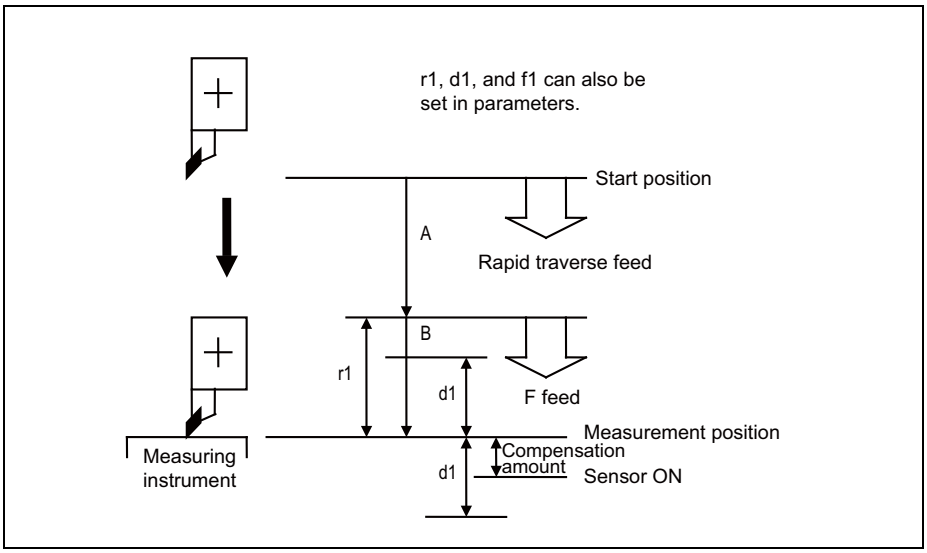
Automatic tool length measurement (L system)

This function moves the tool in the direction of the tool measurement position by commanding a distance between the measurement start position to the measurement position, it stops the tool as soon as it contacts the sensor and calculates the difference between the coordinate position when the tool has stopped and commanded coordinate position. It registers this difference as the tool length compensation amount for that tool.
If compensation has already been applied to the tool, it is moved in the direction of the measurement position with the compensation still applied, and when the measurement and calculation results are such that a further compensation amount is to be provided, the current wear compensation amount is further corrected.

G37 αα1 Rr1 Dd1 Ff1 ;

G37	: Measurement command
αα1	: Measurement axis address and measurement position coordinate (X, Z)
Rr1	: The distance between the point at which tool movement is to start at the measurement speed and the measurement position (Radial value fixed/incremental position)
Dd1	: The range in which the tool is to stop (Radial value fixed: incremental position)
Ff1	: The measurement rate

When Rr1_, Dd1_ and Ff1_ have been omitted, the values set in the parameters are used.



When the tool moves from the measurement start position to the measurement position specified in G37 x1 (z1), it passes through the A area at rapid traverse. Then, it moves at the measurement rate set in F command or parameter from the position specified in r1. If the measurement position arrival signal (sensor signal) turns ON during the tool is moving in the B area, an error occurs. If the measurement position arrival signal (sensor signal) does not turn ON although the tool passes through the measurement position x1 (z1) and moves by d1, an error occurs.

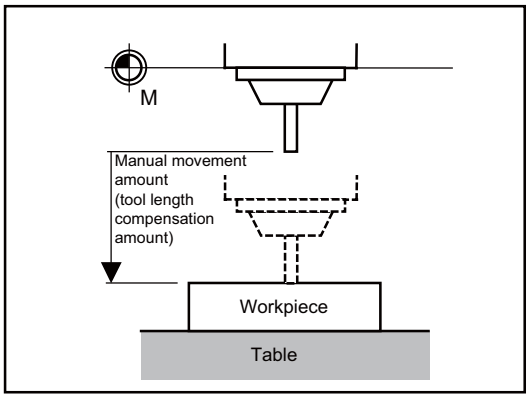
14.1.3 Manual Tool Length Measurement 1

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

Simple measurement of the tool length is done without a sensor.

Manual tool length measurement 1 [M system]

When the tool is at the reference position, this function enables the distance from the tool tip to the measurement position (top of workpiece) to be measured and registered as the tool length compensation amount.

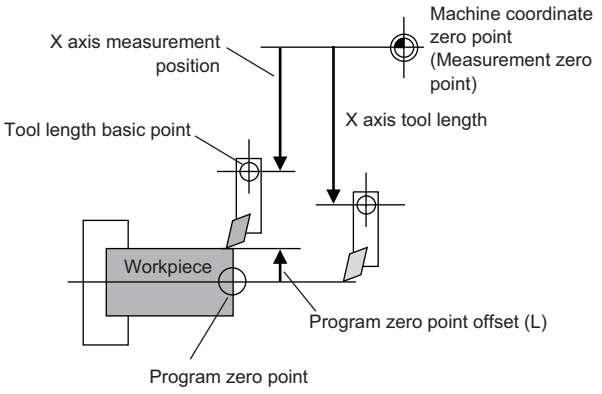


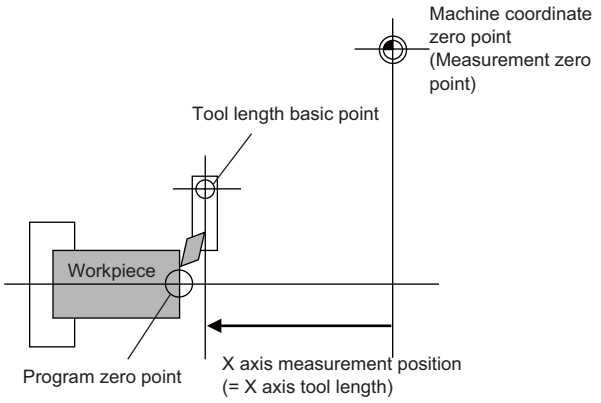
Manual tool length measurement 1 [L system]

This is the function to calculate the tool length compensation amount automatically by moving the tool to the measurement point with manual feed. There are three measurement methods in manual tool length measurement 1, and the value to be specified as measurement zero point differs. The method is selected depending on the parameter settings.
For key operation there are normal operation mode/simple operation mode.

■ Machine zero point designation method

In this method, the measurement zero point is the machine coordinate system zero point.
The tool is moved to the end face or side face, and the tool compensation amount is obtained from the measurement position (machine position) and the program zero point offset (L).

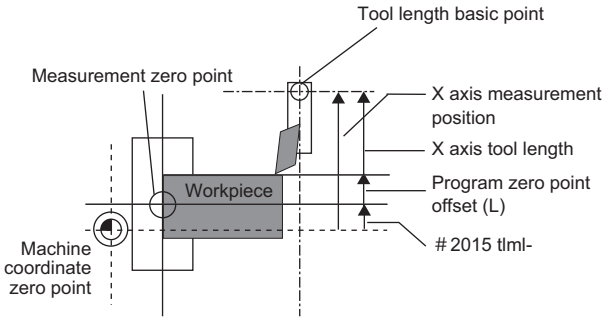


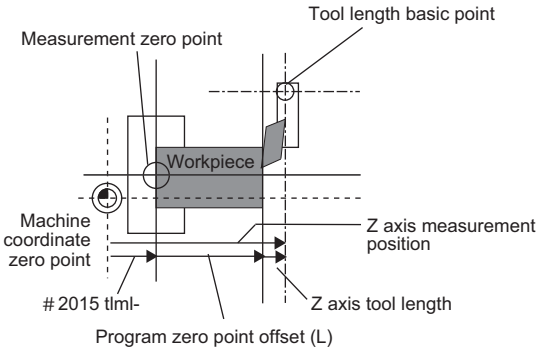


The tool length is automatically calculated by the equation below.
Tool length = Measurement position - Program zero point offset (L)

■ Parameter designation method

In this method, the measurement zero point is the coordinate set to "#2015 tlml-". The offset amount from the machine coordinate zero point to the measurement zero point is set to "#2015 tlml-".
The tool is moved to the end face or side face, and the tool compensation amount is obtained from the measurement position (machine position - measurement zero point (#2015 tlml-)) and the program zero point offset (L).





The tool length is automatically calculated by the equation below.
Tool length = Measurement position (Machine coordinate value - Measurement zero point (#2015 tlml-)) - Program zero point offset (L)

■ Workpiece coordinate offset designation method

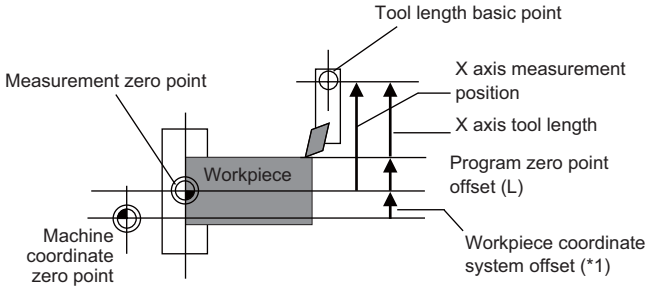
In this method, the measurement zero point can be selected from the following depending on the parameter setting.

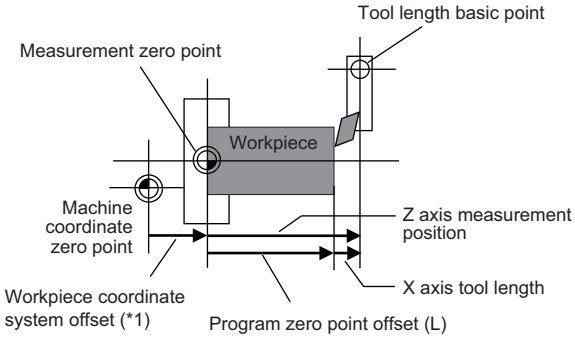
(1) The measurement zero point is the position shifted by the amount of the selected workpiece coordinate system offset from the machine coordinate zero point.

(2) The measurement zero point is the position shifted by the total amount of the following offsets from the machine coordinate zero point.

- Selected workpiece coordinate system offset
- External workpiece coordinate offset
- Workpiece coordinate system shift
- Workpiece base point offset

The tool is moved to the end face or side face, and the tool compensation amount is obtained from the measurement position (workpiece coordinate position) and the program zero point offset (L).





(*1) In the above (2), 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 equation below.

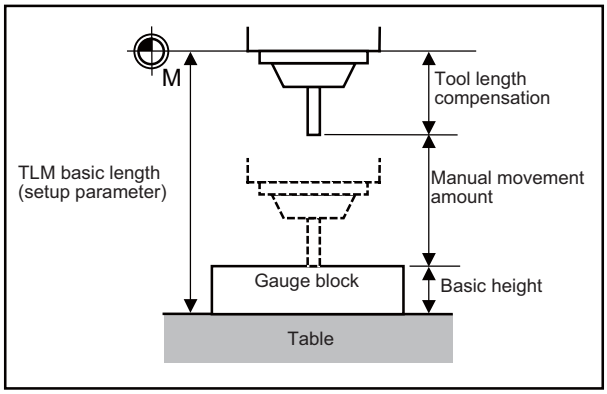
Tool length = Measurement position (Machine coordinate value - Measurement zero point) - Program zero point offset (L)

14.1.4 Manual Tool Length Measurement 2

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

Manual tool length measurement 2 [M system]

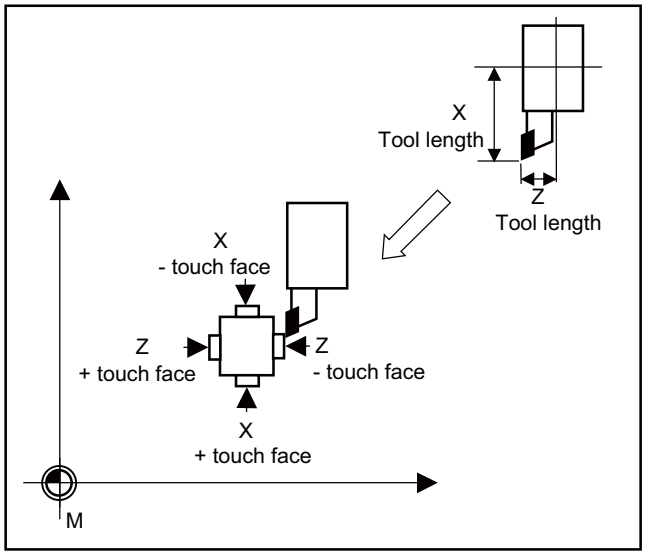
When the tool is positioned at the reference position, this function enables the distance from the reference position to the tool tip to be measured and registered as the tool length compensation amount. In this case, the position of the gauge block used as a reference must be set as the basic height.



If the height axis designation parameter is ON, the axis designated for plane selection basic axis K is the axis targeted for measurement as the height axis.
Furthermore, if the tool length measurement check parameter is ON, an input OK/cancel confirmation message appears after input key has been pressed.

Manual tool length measurement 2 [L system]

A device in which a touch probe is built is used. Simply by causing the tool nose to touch the touch probe in manual feed, the tool compensation amount can be calculated and stored in tool length compensation amount memory. If you set the tool compensation values of each tool, cut the workpiece end face manually, and then input the "Workpiece measurement" signal, you can set the external workpiece coordinate offset data of Z axis.
Preset the machine coordinate position of the touch probe touch face in parameter as the measurement basic position.



14.1.5 Workpiece Coordinate Offset Measurement

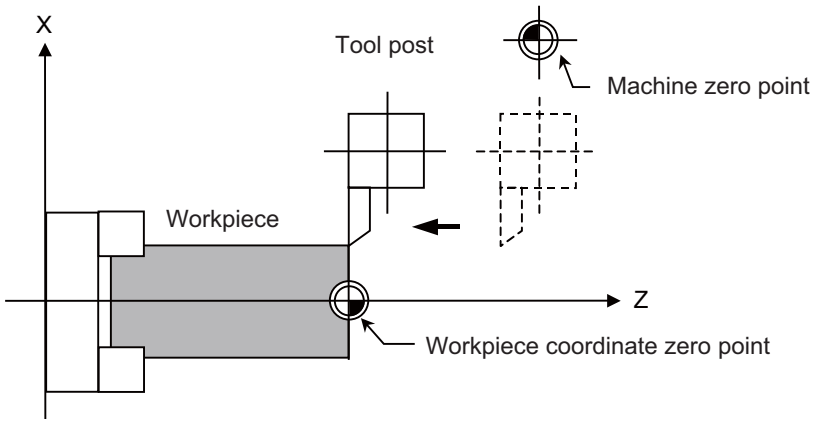
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	O	O	O	O	O	O	O

The external workpiece coordinate offset data for the Z axis can be set by cutting the workpiece face by means of manual operations and inputting the "Workpiece measurement" signal.
By pressing the menu key, data can be set in the Z axis of an arbitrary coordinate offset.
Coordinate offset setting for the Z axis (2nd axis) of the part system 1 to 4 is possible.
Note that the workpiece coordinate offsets for multiple part systems cannot be measured at the same time.

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



[Measurement method with "Workpiece measurement" signal]

- Select a tool, and cut the workpiece face.
- When the "Workpiece measurement" signal is input, the external workpiece coordinate offset data for the Z axis is calculated from the machine coordinate position, length of the tool used and tool nose wear compensation amount, and stored in the memory.

[Measurement method with "MeasVal take in" menu]

- Select a tool, and cut the workpiece face.
- When the "MeasVal take in" menu is pressed, workpiece coordinate offset data is calculated from the machine coordinate position, tool length and tool nose wear compensation amount of a used tool, and external workpiece coordinate offset. The workpiece coordinate offset data is stored in the Z-axis of the selected coordinate system offset.

14.1.6 Workpiece Position Measurement

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	—	—	—	—	—	—	—

The workpiece position measurement function is used to measure each axis' coordinate point by installing a sensor on the spindle and the sensor contacting the workpiece with the manual feed or handle feed.
The surface, hole center and width center coordinates are calculated from the measured coordinates, and those calculated results are set in the workpiece coordinate offset.
The axis to be measured is designated with parameter.
The workpiece position measurement is available for all the part systems.

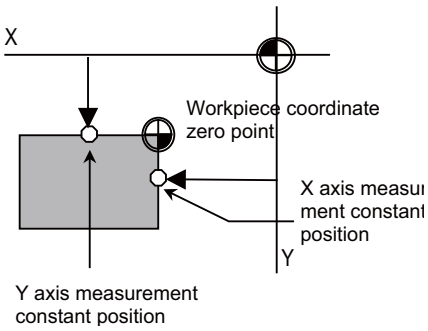
Here the measurement axes are explained as "X", "Y" and "Z".

Surface workpiece offset measurement

The workpiece position measurement coordinates are calculated from the skip machine position of the X, Y and Z axes.
Measurement position coordinate X = X axis' skip machine position + sensor diameter/2 ± center compensation amount (horizontal) ± skip flow amount (horizontal)
Measurement position coordinate Y = Y axis' skip machine position + sensor diameter/2 ± center compensation amount (vertical) ± skip flow amount (vertical)
Measurement position coordinate Z = Z axis' skip machine position - sensor length

The sensor diameter/2 and skip flow amount change between +/- with the tool movement direction during the measurement.
The X and Y axes use the sensor diameter, center compensation amount (horizontal/vertical) and skip flow amount (horizontal/vertical).
The Z axis uses the sensor length.

The measurement position coordinate of the X axis, Y axis or Z axis is set in the specified workpiece coordinate offset.

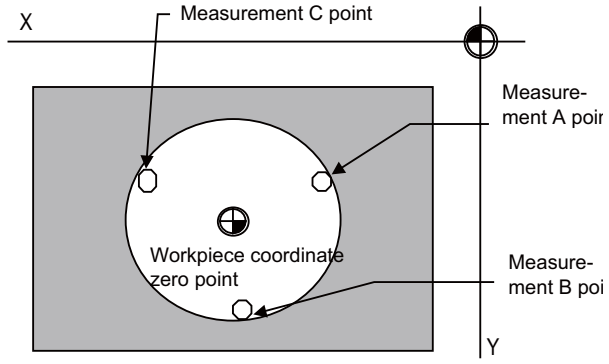


To set the workpiece coordinate offset, the X axis is measured and the X axis' offset coordinate is set.
Then, the Y axis' offset is measured and set.
Finally, the Z axis' offset is measured and set.

Hole center workpiece offset measurement

The measurement position coordinates of two axes (X, Y) are measured at three points, and the hole center is calculated. The calculated result is set in the specified workpiece coordinate offset.
The workpiece position measurement coordinates are calculated from the skip machine position of the X and Y axes.
Measurement position coordinate X = X axis' skip machine position + center compensation amount (horizontal) ± skip flow amount (horizontal)
Measurement position coordinate Y = Y axis' skip machine position + center compensation amount (vertical) ± skip flow amount (vertical)

The +/- of the skip flow amount changes according to the tool movement amount during measurement.

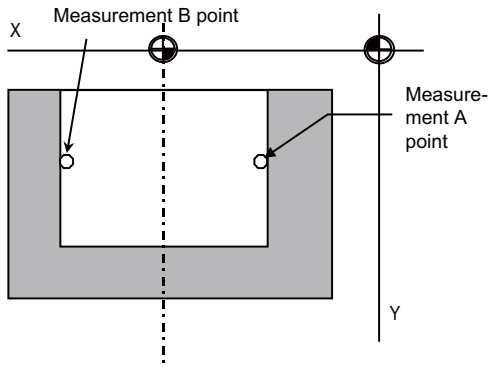


To set the workpiece coordinate offset, the position X and Y of the measurement A point are measured, and the measured values are set in the measurement A point. In the same manner as the measurement A point, the measurement B point and then C point are measured and set. The hole center coordinate is calculated by setting the workpiece coordinate system after setting three points, and the calculated result is set in the workpiece coordinate offset.

Width center workpiece offset measurement

The two points among the measurement position coordinate of the X axis, Y axis or Z axis are measured, and each axis' groove center is calculated. The calculated result is set in the specified workpiece coordinate offset.
The workpiece position measurement coordinates are calculated from the skip machine position of the X, Y and Z axes.
Measurement position coordinate X = X axis' skip machine position + center compensation amount (horizontal) ± skip flow amount (horizontal)
Measurement position coordinate Y = Y axis' skip machine position + center compensation amount (vertical) ± skip flow amount (vertical)
Measurement position coordinate Z = Z axis' skip machine position - sensor length

The +/- of the skip flow amount changes according to the tool movement amount during measurement.



To set the workpiece coordinate offset, the position X (position Y, position Z) of the measurement A point is measured, and the measured value is set in the measurement A point. In the same manner as the measurement A point, the measurement B point is measured and set.
The groove width center coordinate of the X axis (Y axis, Z axis) is calculated by setting the workpiece coordinate system after setting two points, and the calculated result is set in the workpiece coordinate offset

14.1.7 Rotation Measurement

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	—	—	—	—	—	—	—

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.

[Measurement using touch probe]
Measurement counter X = X axis skip position (Machine position)
Measurement counter Y = Y axis skip position (Machine position)

[Simple measurement (measurement without using touch probe)]
Measurement counter X = X axis machine position + OFFSET X + Skip past amount(H) (*1)
Measurement counter Y = Y axis machine position + OFFSET Y + Skip past amount(V) (*1)

(*1) The skip past amount is added for only the axis that moved last.
The sign (+ or -) of the skip past amount depends on the movement direction of the axis.

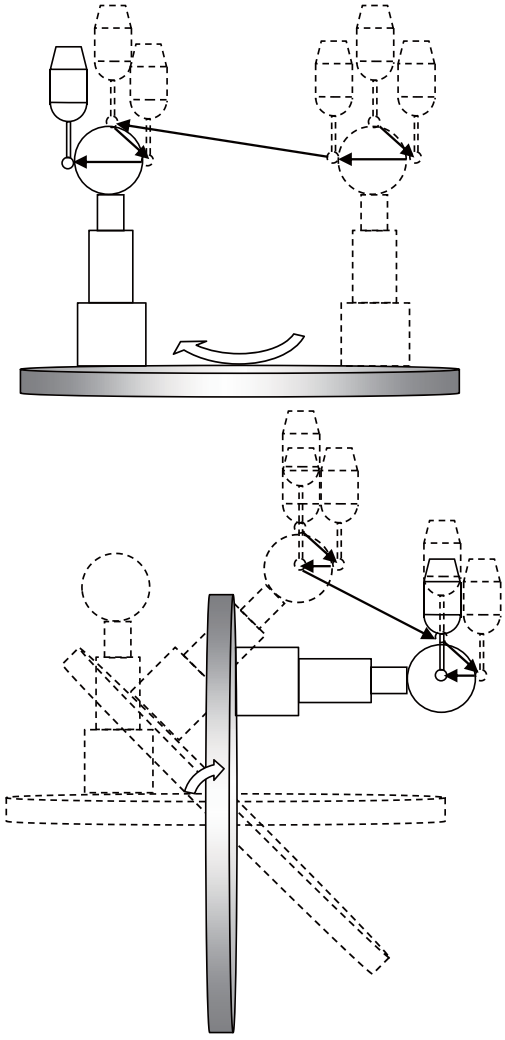
14.1.8 Rotation Center Error Measurement

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	—
L	—	—	—	—	—	—	—

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.
As the movement of the touch probe, the rotation of the axis, the reading of the coordinate and the calculation of the error are performed automatically, the rotation center error compensation amount can be easily calculated. The calculated error compensation amount can be confirmed on the rotation center error measurement screen and set to the parameter.
In order to perform the rotation center error measurement, you need to execute the rotation center error macro initialization in advance and input the rotation center error measurement macro program to the memory.

This function supports the following machine configurations only:
- Table tilt type (A-C axes)
- Table tilt type (B-C axes)
- Combined type (A-C axes)
- Combined type (B-C axes)

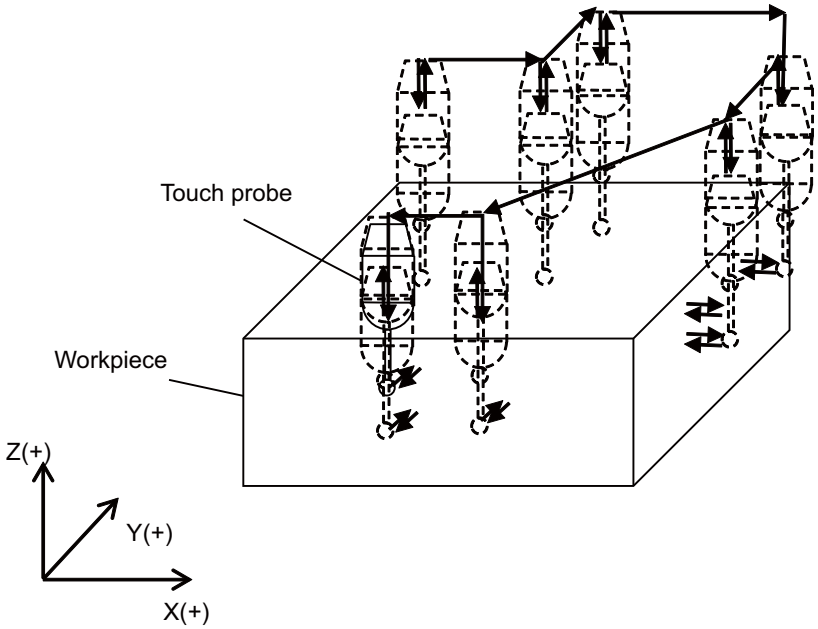
Table tilt type: A machine with two rotary axes on table side
Combined type: A machine with a rotary axis each on tool head side and table side
If the rotation center error measurement is attempted with the configuration which is not subject, the alarm occurs when the automatic start button is pressed.



14.1.9 Workpiece Installation Error Measurement

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	—	Δ	—	—	—	—
L	—	—	—	—	—	—	—

This function executes automatic measurement with the touch probe to calculate the workpiece installation error compensation amount.
As the movement of the touch probe, the reading of the coordinate and the calculation of the error are performed automatically, the workpiece installation error compensation amount can be easily calculated. The calculated error compensation amount can be confirmed on the workpiece installation error measurement screen and set to the error compensation amount.
In order to perform the workpiece installation error measurement, you need to execute the workpiece installation error macro initialization in advance and input the workpiece installation error measurement macro program to the memory.



14.2 Tool Life Management

14.2.1 Tool Life Management

Tool life management is a function to monitor the usage of the programmed tool based on the cumulative cutting hours or the cumulative number of cuttings.
The function enables appropriate exchange of tools, which leads to consistent quality of machining.
For multi-part system, the number of tools for tool life management can be set individually for each part system.

14.2.1.1 Tool Life Management I

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

The following two method can be selected.

- Management by the cutting hours
  The cutting time (such as G01, G02, or G33) after tool selection (T) command is added to the cumulative cutting hours for the specified tool.
- Management by the number of cuttings
  The cumulative number of cuttings of the specified tool is incremented each time a tool selection (T) command is issued for the tool.
  The number of times of hole drilling in fixed cycle for drilling can be counted by setting the parameter.

[M system]
The "Tool life over" signal will turn ON when the cumulative tool usage data has reached the end of tool life.

[L system]
The "Tool life over" signal will turn ON when a T command is issued after the cumulative tool usage data has reached the end of tool life.

14.2.1.2 Tool Life Management II

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

[M system]
A spare tool change function is added to tool life management I. This function selects a usable tool out of the spare tools of the group determined by the value specified by the user PLC, then outputs data of such usable spare tool. The spare tool can be selected in two ways: the tools are selected in order they were registered in the group, or the tool whose remaining period of the tool life is the longest of all in the group is selected.

[L system]
The tool life of each tool (cutting hours or number of cuttings) is controlled, and when the usage data has reached the end of tool life, a spare tool of the same type is selected from the group where the tool belongs to.
By setting the remaining tool life value prediction value (hours or the number of times) in advance, when the remaining tool life of the tool in use is below the set value, the "Tool life prediction" signal can be output. (Tool life prediction function)

14.2.1.3 Tool Life Management III

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	—	—	—	—	—	—	—

For the tools designated by user PLC, tools' cumulative cutting hours (0 to 4000 minutes) or cumulative number of cuttings (0 to 65000 times) is accumulated, and the tool usage state is monitored. The tool life data for up to 1000 tools can be managed. This function does not support the management by the group No.

14.2.2 Number of Tool Life Management Sets

[M system]

Number of tool life management sets	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
Number of tool life management tools (99 sets)	—	—	—	—	—	—	—
Number of tool life management tools (128 sets)	—	—	—	—	—	—	—
Number of tool life management tools (200 sets)	○	○	○	○	○	○	○
Number of tool life management tools (256 sets)	—	—	—	—	—	—	—
Number of tool life management tools (400 sets)	△	△	△	△	—	—	—
Number of tool life management tools (999 sets)	△	△	△	△	—	—	—

[L system]

Number of tool life management sets	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
Number of tool life management tools (99 sets)	—	—	—	—	—	—	○
Number of tool life management tools (128 sets)	○	○	○	○	—	—	—
Number of tool life management tools (200 sets)	—	—	—	—	—	—	—
Number of tool life management tools (256 sets)	—	—	—	—	○	○	—
Number of tool life management tools (400 sets)	△	△	△	△	—	—	—
Number of tool life management tools (999 sets)	△	△	△	△	—	—	—

The number of tools that can be managed for their lives in the case of fixed allocation are shown below. For details on the arbitrary allocation, refer to " 14.2.3 Tool Life Management Set Allocation to Part Systems".

[M system]

Function name	Common for part systems or 1-part system	Independent to each part system and multi-part system
Number of tool life management tools (200 sets)	200 tools	Divide the number of tools in the system by the number of part systems to calculate the number of tools in each part system(If there is the remainder, the remainder is allocated to the first part system.)
Number of tool life management tools (400 sets)	400 tools	
Number of tool life management tools (999 sets)	999 tools	

[L system]

Function name	Common for part systems or 1-part system	Independent to each part system and multi-part system
Number of tool life management tools (99 sets)	99 tools	Divide the number of tools in the system by the number of part systems to calculate the number of tools in each part system(If there is the remainder, the remainder is allocated to the first part system.)
Number of tool life management tools (128 sets)	128 tools	
Number of tool life management tools (256 sets)	256 tools	
Number of tool life management tools (400 sets)	400 tools	
Number of tool life management tools (999 sets)	999 tools	

14.2.3 Tool Life Management Set Allocation to Part Systems

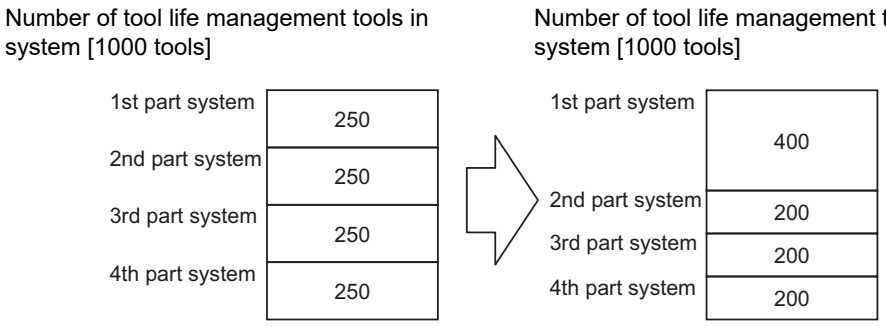
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

(*) Variable number of management tools

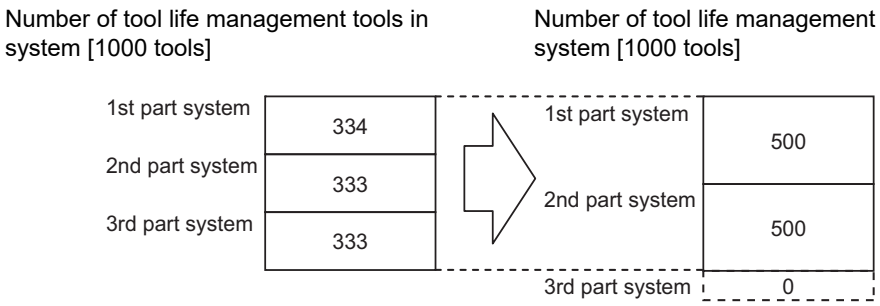
The number of tool life management tools can be set per part system.
There are two types of the allocation: "Arbitrary allocation" which allocates the number of tool life management tools to each part system arbitrarily and "Fixed allocation" which automatically allocates the number of tool life management tools to each part system equally, and the type can be selected using the parameter.
The arbitrary allocation enables the efficient allocation because when a certain part system needs only a small number of tool life management tools, the rest can be allocated to another part system. If an auxiliary-axis part system does not need the tool life management at all, the number of tool life management tools can be set 0 in the auxiliary-axis part system.
Number of tool life management tools in system is the total number of tool life management tools of all part systems.

■ Arbitrary allocation

[When the number of tool life management tools is increased in the 1st part system of 4-part system]

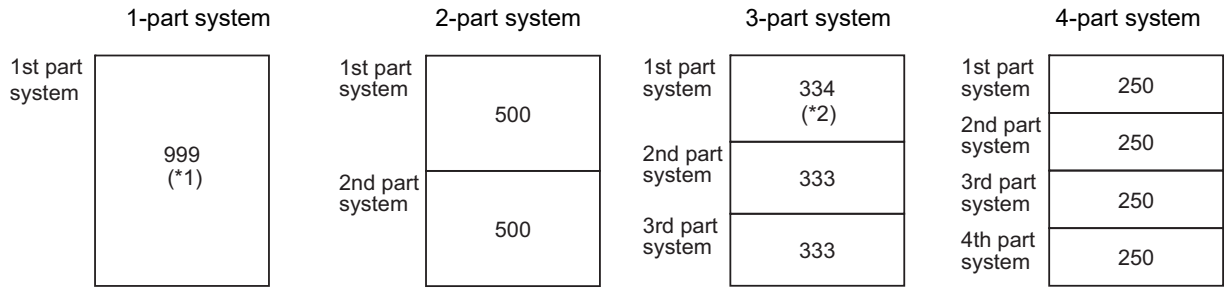


[When the number of tool life management tools is set "0" to the 3rd part system of 3-part system for use of the 3rd part system as auxiliary-axis part system]



■ Fixed allocation

Number of tool life management tools in system [1000 tools]



(*1) The maximum number of tool life management tools for each part system is 999.

(*2) If there is any remainder, the remainder is allocated to the 1st part system.

14.3 Others

14.3.1 Programmable Current Limitation

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

This function allows the current limit value of the NC axis to be changed to a desired value in the program, and is used for the workpiece stopper, etc.
The commanded current limit value is designated with a ratio of the limit current to the rated current. The current limit value can also be set from the setting and display unit.
The validity of the current limit can be selected with the external signal input.
However, the current limit value of the PLC axis cannot be rewritten.

G10 L14 X dn ;

G10	: Current limit input command
L14	: Current limit value setting (+ side/- side)
X	: Axis address
dn	: Current limit value 1 % to 999 %

- If the current limit is reached when the current limit is valid, the "Current limit reached" signal is output.
- The following two modes can be used with external signals as the operation after the current limit is reached.
  - Normal mode
    The movement command is executed in the current state.
    During automatic operation, the movement command is executed to the end, and then the next block is moved to with the droops still accumulated.
  - Interlock mode
    The movement command is blocked (internal interlock).
    During automatic operation, the operation stops at the corresponding block, and the next block is not moved to.
    During manual operation, the following same direction commands are ignored.
- During the current limit, the droop generated by the current limit can be canceled with external signals.
  (Note that the axis must not be moving.)
- The setting range of the current limit value is 1% to 999%. Commands that exceed this range will cause a program error.
- If a decimal point is designated with the G10 command, only the integer will be valid.
  (Example) G10 L14 X10.123 ; The current limit value will be set to 10%.
- For the axis name "C", the current limit value cannot be set from the program (G10 command).
  To set from the program, set the axis address with an incremental axis name, or set the axis name to one other than "C".

14.3.2 Auto Power OFF

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

This can notify that the control unit's power can be turned OFF after shutting the display unit down (Do not turn the power OFF.) by entering "Automatic power OFF request" signal from user PLC to NC.

Note

(1) The Mitsubishi Electric-supplied display unit is required.

14.3.4 Load Monitoring I

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	—
L	Δ	Δ	Δ	Δ	○	○	—

This function aims at detecting tool wear or degradation by detecting and monitoring the actual load (current value) on spindle and NC axes on a real time basis.
The effective torques and estimated disturbance torques on the spindle and NC axes are output to the predetermined R registers in real time. Sampling of the estimated disturbance torque can be performed with this function.
Up to 8 servo axes in each part system (maximum 4 part systems) are supported with this function.

14.3.5 Power ON/OFF Sequence

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	—	—	○	—	—
L	○	○	—	—	○	—	—

This function turns the power supply ON/OFF, synchronizing the NC control unit and the HMI screen.
This enables the following:

- The relay which synchronizes the control unit and the personal computer unit when the power supply is turned ON becomes unnecessary.
- The control unit and the personal computer unit are configured with the same power supply and can be started up at optimum timing even when they are turned ON at the same time.
- By using the power ON/OFF button of the operation panel side, both the control unit and the personal computer unit can be started up and their power supply can be turned ON again.
- When the automatic power OFF is performed, NC starts the end processing right after the end processing of the personal computer unit is completed to reduce the wasteful standby time.

14.3.7 Direct Robot Control

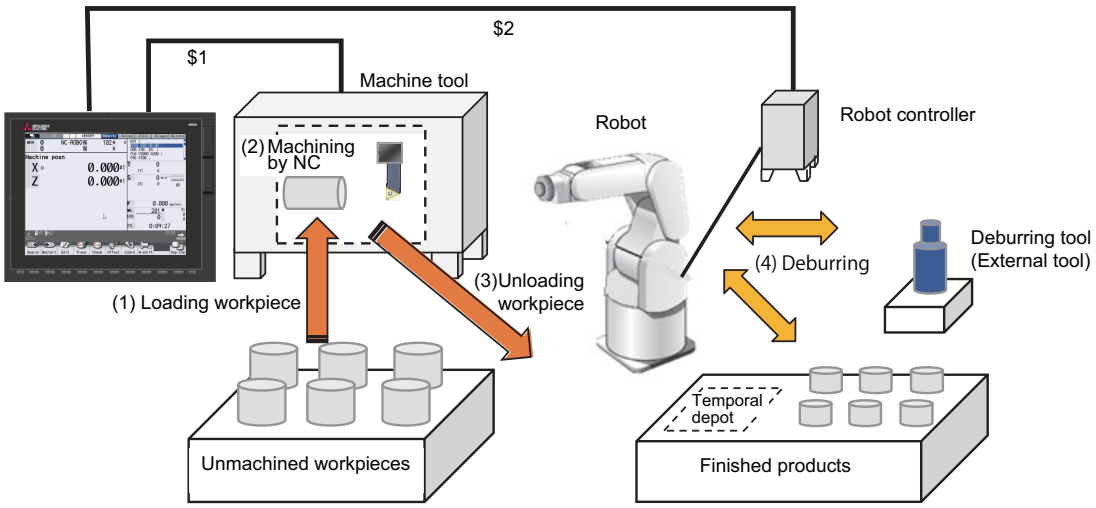
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	□	□	□	□	□
L	□	□	□	□	□	□	□

This function enables a robot to be connected to and directly operated by an NC.
A robot-dedicated screen is provided to display the status of the robot on the NC screen. Some of the possible operations are as follows:

- Automatic operation using robot-dedicated G code commands in NC machining programs
- Manual operation of the robot using a handle on the NC
- Digital I/O control to open and close the robot hand

As in the following example, this function enables the NC to control the machining of workpieces with the robot loading and unloading workpieces in coordination with the machining, as well as deburring through the robot concurrent with the machining of workpieces.

[Example of system configuration]
1st part system ($1) : X, Z
2nd part system ($2) : - (The part system has no axes because only a robot is controlled.)

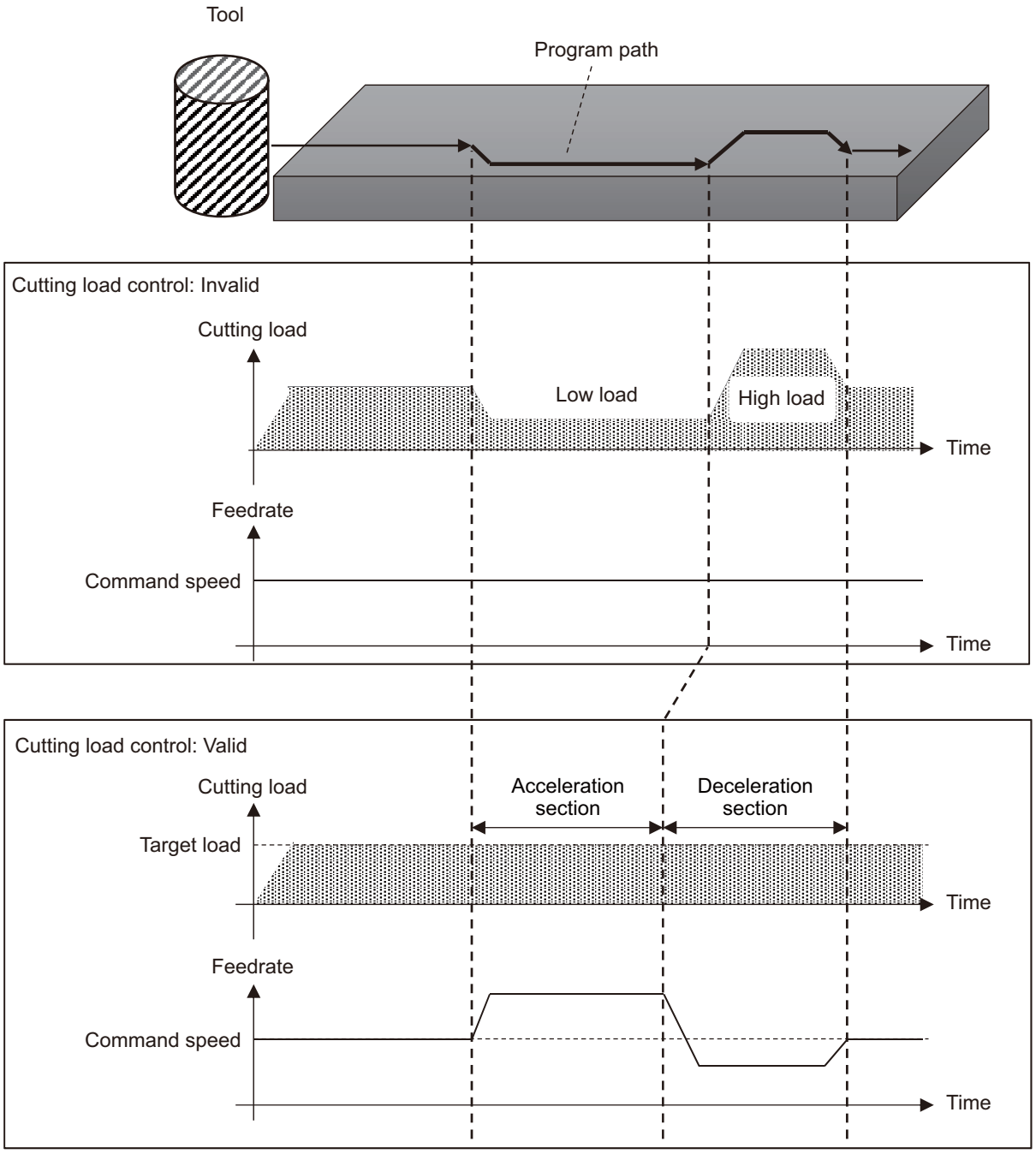


A part system to be used exclusively for robot operation (robot-dedicated part system) can be set. Even when the robot-dedicated part system is not used, robot operation is possible; however, when the robot-dedicated part system is used, robot operation is possible with the same command (program) as the NC axis control.

14.3.8 Cutting Load Control

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	—
L	—	—	—	—	—	—	—

This function automatically changes the feedrate so that the cutting load reaches the set target load. For machining with a low cutting load, the feedrate can be increased and the cycle time can be shortened. On the other hand, for machining with a high cutting load, the tool can be protected by reducing the feedrate and reducing the load on the tool.
This function may affect the surface quality of the workpiece as the feedrate increase or decrease; therefore, use it for the machining such as rough machining.

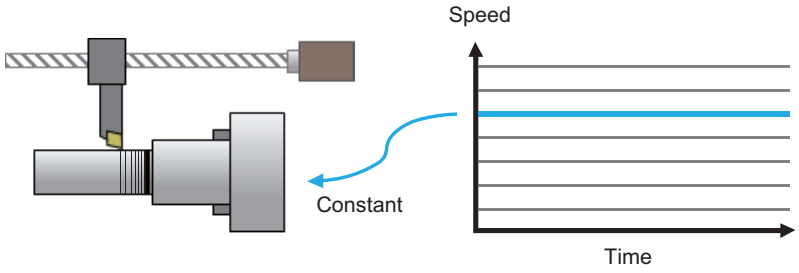


14.3.9 Chatter Suppression

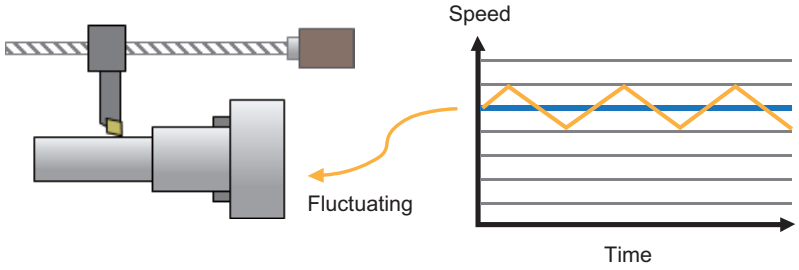
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	△	△	△	△	○	○	—

Chatter is vibration which occurs continuously between tool and workpiece during the turning machining. This function suppresses the vibration by periodic fluctuation of spindle speed.
This function applies a triangle wave of designated amplitude and cycle to the spindle and fluctuates the speed, which reduces machining defects caused by the chatter.

[Normal case]



[Chatter suppression]



15

Safety and Maintenance

15.1 Safety Switches

15.1.1 Emergency Stop

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

All operations are stopped by the emergency stop signal input and, at the same time, the drive section is stopped and the movement of the machine is stopped.
At this time, the READY lamp on the setting and display unit goes OFF and the "Servo ready completion" signal is turned OFF.
When the emergency stop is cancelled, the reset will be performed. The reset type can be selected in the parameter.

15.1.2 Data Protection Key

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

The editing functions such as setting and deleting of data can be prohibited by the data protection key. There are 5 types of data protection keys, and each data protection key and the data to be protected are as follows:

Data protection key type	Details
Data protection key 1	For protecting the overall tool data and the coordinate system presettings as based on origin setting.
Data protection key 2	For protecting the user parameters and common variables.
Data protection key 3	For protecting the machining programs of NC memory and NC memory 2.
Data protection key (memory card)	For protecting the data on the front-side SD (memory card).
Data protection key (DS)	For protecting the data on the back-side SD (DS).

The data protection keys prohibit the editing functions such as setting and deleting when the keys are OFF (0).

15.2 Display for Ensuring Safety

15.2.1 NC Warning

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

The warnings which are output by the NC system are listed below. When one of these warnings has occurred, a warning number is output to the PLC and a description of the warning appears on the screen. Operation can be continued without taking further action.

Type of warning	Outline
Servo warning	The servo warning is displayed.
Spindle warning	The spindle warning is displayed.
System warning	The system warning is displayed. (State such as temperature rise, battery voltage low, etc.)
Absolute position warning	A warning in the absolute position detection system is displayed.
Auxiliary axis warning	The auxiliary axis warning is displayed.

15.2.2 NC Alarm

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

When an alarm has occurred, an alarm number is output to the PLC, and a description of the alarm appears on the screen. Operation cannot be continued without taking remedial action.

For NC alarm display and alarm signals, refer to "Alarm/Parameter Manual" and "PLC Interface Manual" for details.
In addition, the NC alarm has the following functions.

■ Display delay function of an operation alarm

This function can be used to postpone only alarm display (including alarm signal ON and alarm history record) when an operation error has occurred.
By removing the causes of the operation error while alarm display is being postponed, unnecessary alarm display is not displayed and also unnecessary alarm history is not recorded.

■ 7-segment LED alarm/status display

The 7-segment LED mounted on the operation board I/O can display NC status.
When alarms occur, an alarm with the highest priority within each part system is displayed.

■ Alarm/warning on PLC output of servo/spindle drive unit

When an alarm or a warning has occurred in the servo drive unit or spindle drive unit, alarm No. and warning No. of each drive unit is output to the R register.
When a warning has occurred in the servo drive unit or spindle drive unit, the signal is output to indicate that a warning has occurred.

■ Multiple keys cannot be input

This function is used to display an operation error when multiple keys are input with the keyboard or the menu buttons on the display unit.

15.2.3 Operation Stop Cause

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

The stop cause of automatic operation is displayed on the setting and display unit.

15.2.4 Emergency Stop Cause

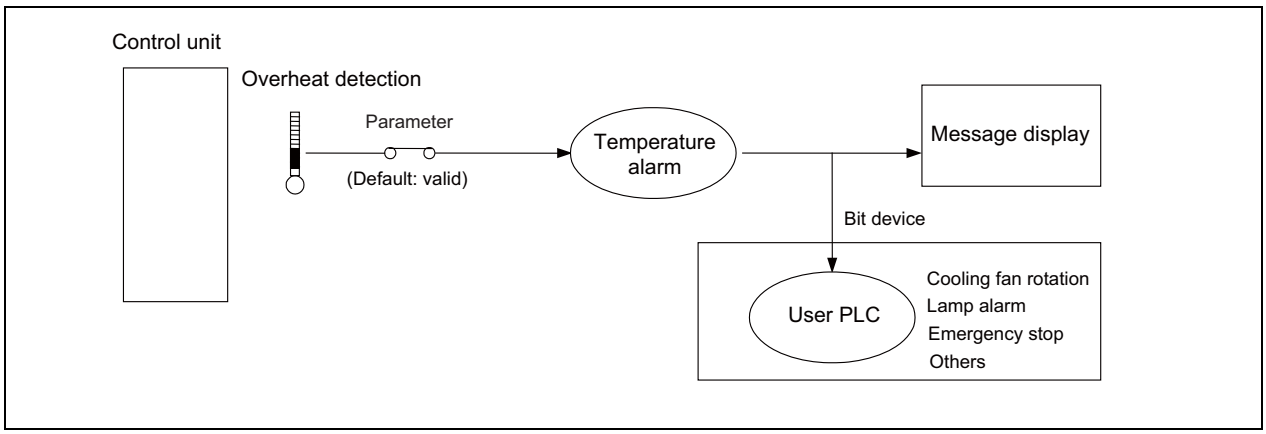
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

When "EMG" (emergency stop) message is displayed in the operation status display area of the setting and display unit, the emergency stop cause can be confirmed.

15.2.5 Thermal Detection

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

When overheating is detected in the control unit, an overheat signal is output at the same time as the alarm is displayed. If the system is in automatic run at the time, run is continued, but it cannot be started after reset or completion by M02/M30. (It can be started after block stop or feed hold.)
When the temperature falls below the specified temperature, the alarm is released and the overheat signal is turned OFF.



Note

(1) If the temperature rise detection function is invalidated with the parameters, the control could be disabled when the temperature is excessive. This could result in machine damage or personal injuries due to runaway axis, and could damage the device. Enable the detection function for normal use.

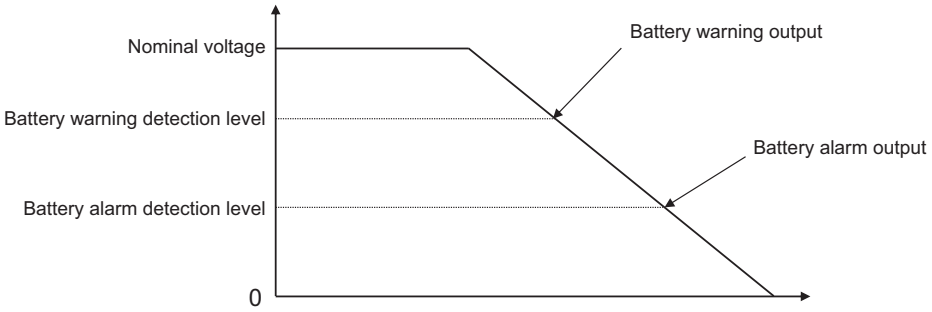
15.2.6 Battery Alarm/Warning

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

When it is time for changing batteries, alarm or warning is displayed.
When a warning is displayed, immediately backup all the necessary data and change batteries.
When an alarm is displayed, there is a possibility that memory has been lost.

The target control devices are as follows.

- Control unit
- Personal computer unit (FCU8-PC232)



15.3 Protection

15.3.1 Stroke End (Over Travel)

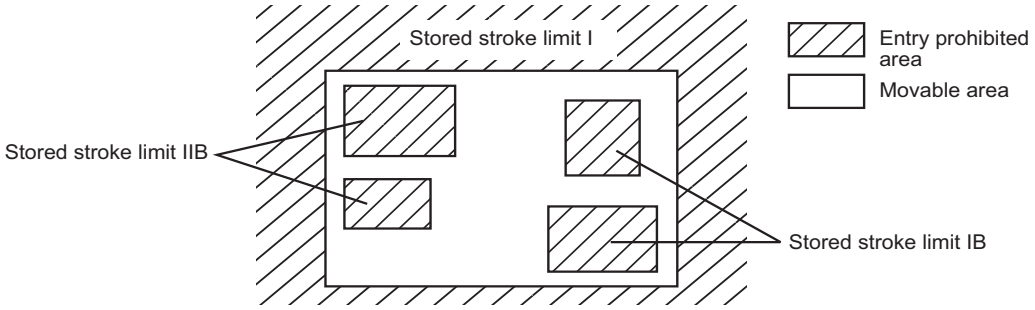
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

When limit switches and dogs have been attached to the machine and a limit switch has kicked a dog, the movement of the machine is stopped by the signal input from the limit switch.
At the same time, the alarm output is sent to the machine.
The stroke end state is maintained and the alarm state is released by feeding the machine in the reverse direction in the manual mode to disengage the dog.

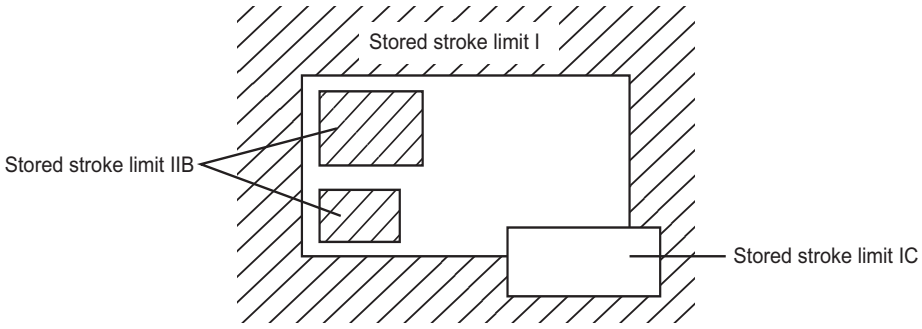
15.3.2 Stored Stroke Limit

The axis can be stopped before entering the prohibited area by setting the axis entry prohibited area.
The inside or outside of the specified area can be set as the entry prohibited area.
There are "stored stroke limit I/II" where the outside of the specified area is the prohibited area, "stored stroke limit IB/IIB" where the inside of the specified range is the prohibited area, and "stored stroke limit IC" where the movable area is specified in the prohibited area.
Up to five entry prohibited areas can be set by combining them.

[Usage example 1]



[Usage example 2]



The axis decelerates to a stop before entering the prohibited area, and an alarm occurs.
The axis that caused the alarm can move only in the opposite direction in which it has just moved.

[Enabling conditions of stored stroke limit]

When the relative position detection system is used, the stored stroke limit is disabled until the reference position return is completed after the power is turned ON.

If the absolute position detection method is set to the simple absolute position detection, the stored stroke limit can be enabled even when the reference position return has not been completed.

Note

(1) If the absolute position detection is enabled when using the absolute position detection system, the stored stroke limit will be enabled 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 after the reference position return is established.

When the stored stroke limit is enabled with the reference position return 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.

The stored stroke limits I, II, IIB, IB and IC are handled as follows.

Type	Area	Details
I	Outside prohibited	• Set by the machine tool builder.
II	Outside prohibited	• Set by the user. • By using the program command, the parameter can be changed and the function can be switched to ON/OFF. • Select II or IIB with the parameters.
IIB	Inside prohibited	
IB	Inside prohibited	• Set by the machine tool builder.
IC	Inside allowed	• Set by the machine tool builder.

Note

(1) The stored stroke limit is disabled for axis where the reference point is not established.

(2) For stored stroke limit I/II, it is disabled for axis where the same values (other than "0") are set to the upper and lower limits of the area setting parameters. The axis where "0" is set to the upper and lower limits of the area setting parameters cannot be moved.

(3) For stored stroke limit IB/IC, the stored stroke limit is disabled for axis where the same values are set to the upper and lower limits of the range setting parameters.

(4) When the stored stroke limit I and II are used together, the overlap of the areas designated by them (I and II) is the movable area.

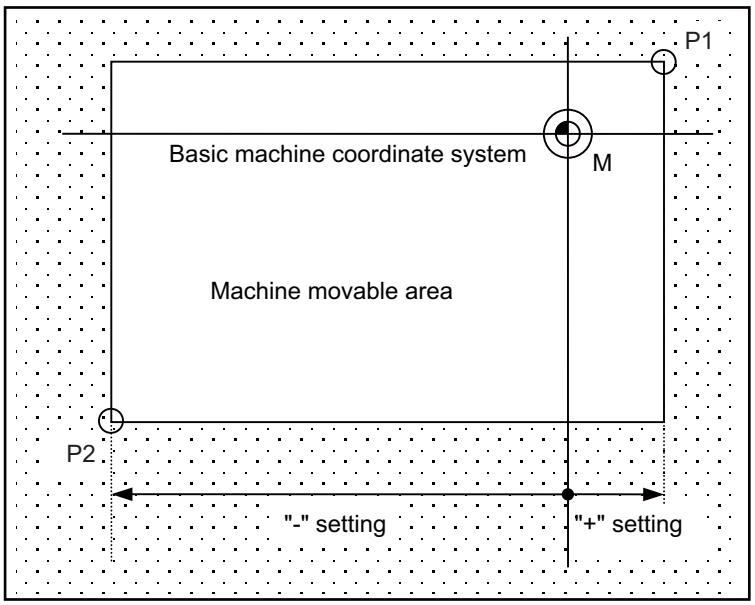
(5) The stored stroke limit IB and IC cannot be used together.

15.3.2.1 Stored Stroke Limit I/II

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

Stored stroke limit I

This is the stroke limit function used by the machine tool builder, and the outside of the boundary is defined as the entrance prohibited area.
The maximum and minimum values for each axis can be set by parameters. The function itself is used together with the stored stroke limit II function described in the following section, and the tolerable area by both functions is the movable area.
The setting range is -99999.999 to +99999.999 mm.



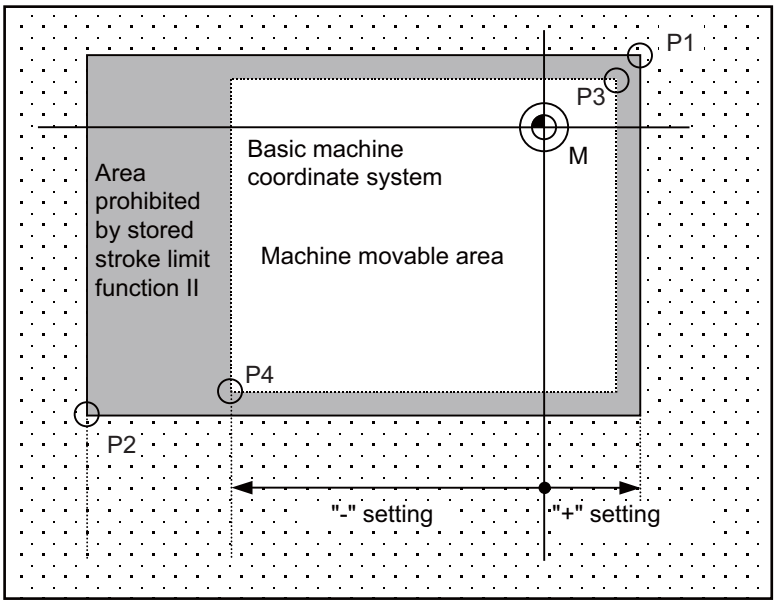
The position of points 1 and 2 are set using the basic machine coordinate.

Prohibited area

The area of the stored stroke limit I can be changed to the value set to R register for each axis.
When "Stored stroke limit I change request" signal is turned ON, the area of stored stroke limit changes. Changing the area for the stored stroke limit I is also possible during automatic operation. Also, the current settings for the stored stroke limit I can be checked with the R register values.
This feature allows an operator to switch the area of the stored stroke limit I during axis movement for the purpose of tool exchange, for example.

Stored stroke limit II

This is the stroke limit function which can be set by the user, and the area outside the set limits is the prohibited area.
The limits for each axis can be set by parameters or with program commands. When this function is used together with the stored stroke limit I function described in the foregoing section, the tolerable area by both functions is the movable area.
The setting range is -99999.999 to +99999.999 mm.
The stored stroke limit (II) function can also be disabled for each axis with the parameter settings.



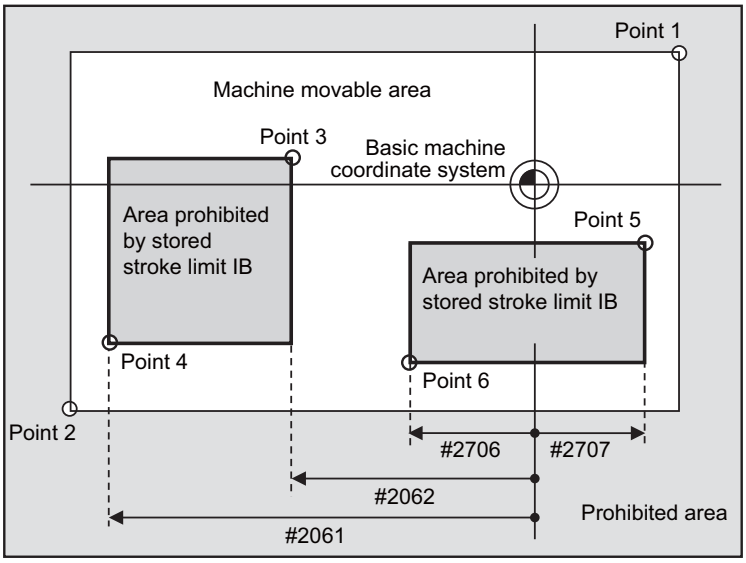
The position of points 3 and 4 are set using the basic machine coordinate.
The area determined by points 1 and 2 is the prohibited area set by stored stroke limit I.

Prohibited area

15.3.2.2 Stored Stroke Limit IB

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

The boundary is set for each axis with parameters. The inside of the set boundary is defined as the prohibited area.
When the stored stroke limit IB is enabled, up to two prohibited areas can be set.

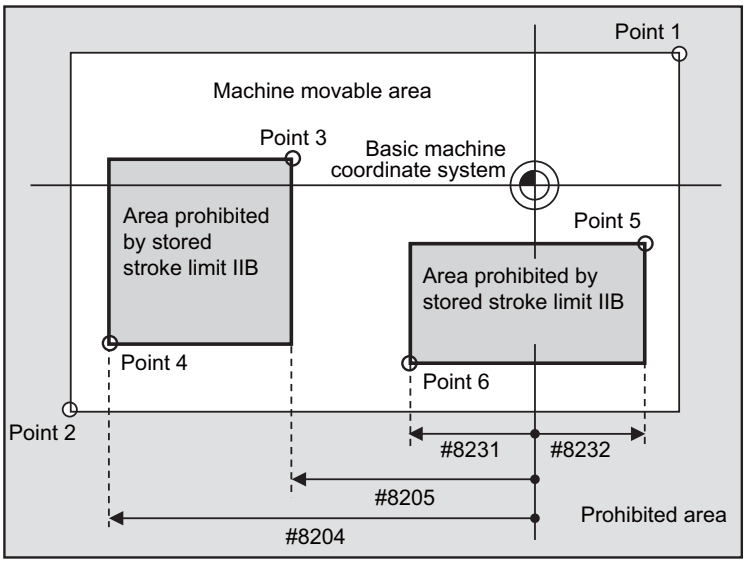


Point 3: #2062 OT_1B+
Point 4: #2061 OT_1B-
Point 5: #2707 OT_1B[2]+
Point 6: #2706 OT_1B[2]-
These values are set with the coordinate position in the basic machine coordinate system.
The area determined by points 1 and 2 is the prohibited area set by stored stroke limit I.

15.3.2.3 Stored Stroke Limit IIB

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

A parameter is used to switch between this function and stored stroke limit II.
With stored stroke limit IIB, the inside of the set boundary is defined as the prohibited area.
When the stored stroke limit IIB is enabled, up to two prohibited areas can be set.
The limits for each axis can be set by parameters or with program commands. However, "#8231 OT-CHECK[2]-N" and "#8232 OT-CHECK[2]-P" cannot be set with program commands.
When the stored stroke limit II is selected, the stored stroke limit IIB is disabled.

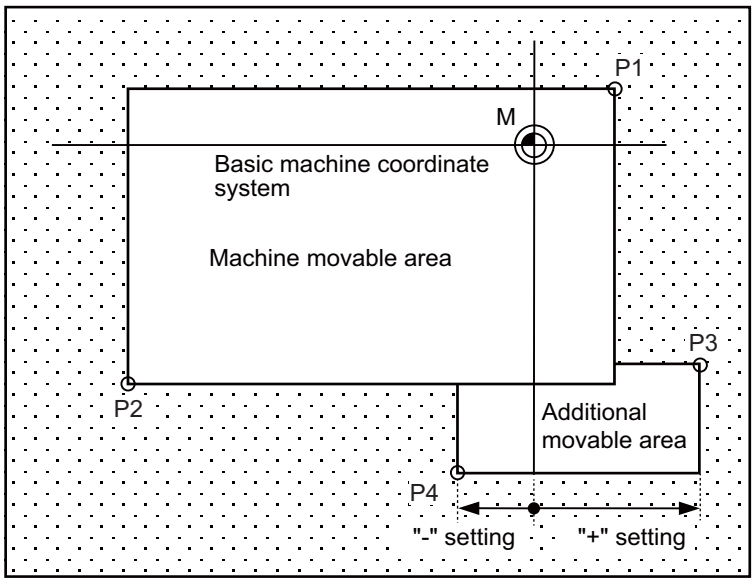


Point 3: #8205 OT-CHECK-P
Point 4: #8204 OT-CHECK-N
Point 5: #8232 OT-CHECK[2]-P
Point 6: #8231 OT-CHECK[2]-N
These values are set with the coordinate position in the basic machine coordinate system.
The area determined by points 1 and 2 is the prohibited area set by stored stroke limit I.

15.3.2.4 Stored Stroke Limit IC

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

The boundary is set for each axis with parameters. The inside of the set boundary is the additional movement range.
This cannot be used with stored stroke limit IB.



The values of points 3 and 4 are set with the coordinate position in the basic machine coordinate system.
The area determined by points 1 and 2 is the prohibited area set by stored stroke limit I.

Prohibited area

15.3.3 Stroke Check Before Travel

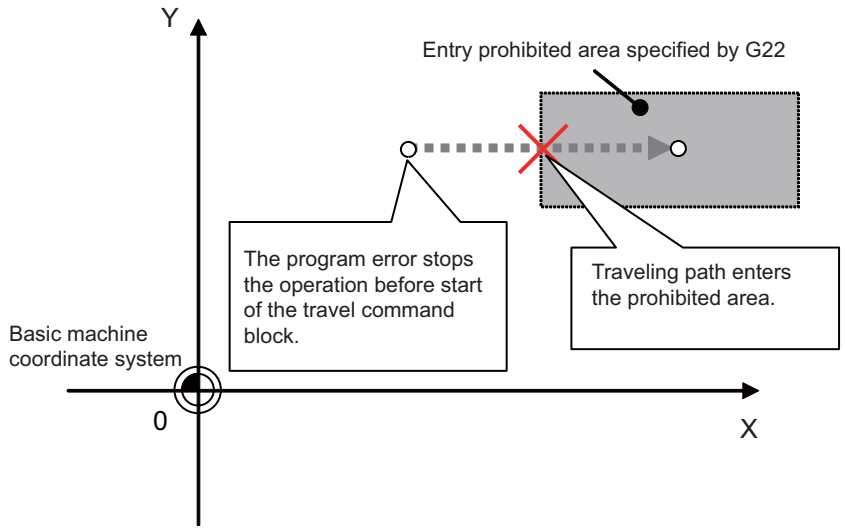
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

This function detects an error of the program command which causes the machine to enter the prohibited area where machine entry is prohibited.
Whereas normal stored stroke limit stops the machine just before the set prohibited area, this function causes the program error to occur in a block where the said command is given, and stops the machine before movement. This facilitates review of program paths as well as continuous operation.

There are two methods for stroke check before travel, which can be selected with a parameter.

Stroke check before travel (G22/G23)

G22 allows you to set the entry prohibited area (X, Y and Z) and enable the stroke check before travel.
The program error occurs in a command block whose start or end point of travel, or linear or circular path enters the prohibited area.
G23 turns OFF the check function.
This method can be used for machining center specification only.



■ Command format

Stroke check before travel ON

G22 X__ Y__ Z__ I__ J__ K__ ;

X, Y, Z (*1)	Coordinates of upper point (basic axis name and its coordinate position)
I,J,K	Coordinates of lower point (I, J, K address and its coordinate position)

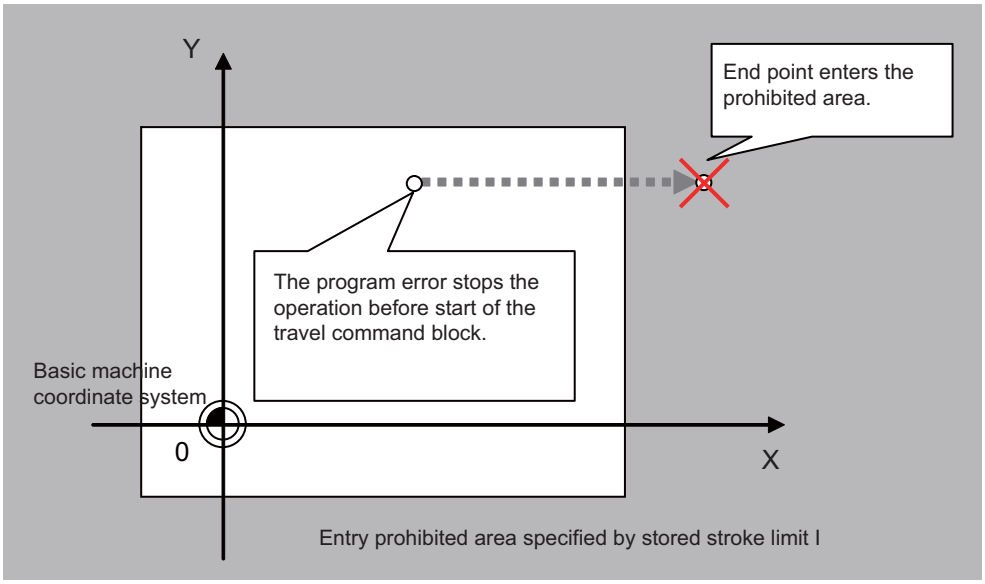
(*1) In the command format, the basic axes are X, Y and Z. Command the basic axis set in the parameter.

Stroke check before travel cancel

G23 ;

Stroke check before travel in the stored stroke limit area

The block in which the end point of the movement command enters the entrance prohibited area set with the stored stroke limit I, IB, IC, II or IIB function becomes a program error.



15.3.4 Chuck/Tailstock Barrier Check

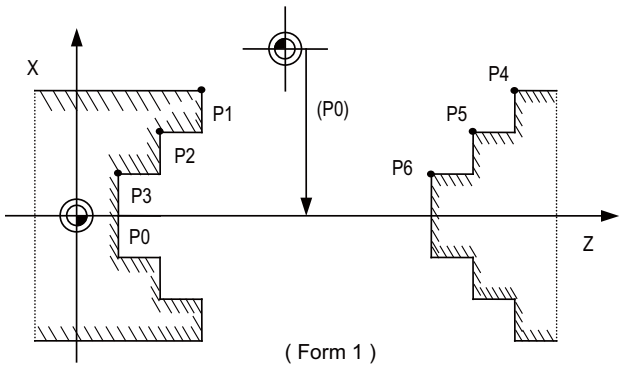
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	O	O	O	O	O	O	O

By limiting the tool nose point move range, this function prevents the tool from colliding with the chuck or tailstock because of a programming error. When a move command exceeding the area set in a given parameter is programmed, the tool is stopped at the barrier boundaries.
Command format

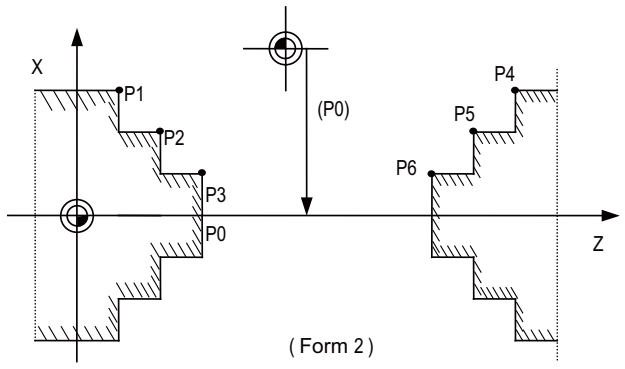
G22 ;	**Barrier ON**

G23 ;	**Barrier OFF (cancel)**

- When the machine is about to exceed the area, the machine is stopped and an alarm is displayed. To cancel the alarm, execute reset.
- The function is also effective when the machine is locked.
- This function is valid when all axes for which a barrier has been set have completed reference position return.
- The chuck barrier/tailstock barrier can be set per each part system.
- Chuck barrier/tailstock barrier setting.



(Form 1)



(Form 2)

The chuck barrier and tailstock barrier are both set with the machine coordinate by inputting one set of three-point data in the parameter.
Points P1, P2 and P3 are the chuck barrier, and points P4, P5 and P6 are the tailstock barrier. The X axis is set with the coordinate position (radius value) from the workpiece center, and the Z axis is set with the basic machine coordinate system coordinate.
Point P0 is the chuck barrier and tailstock barrier's basic X coordinates, and the workpiece center coordinate in the basic machine coordinate system is set.
The barrier area is assumed to be symmetrical for the Z axis, and if the X axis coordinate of barrier point P_ is minus, the sign is inverted to plus and the coordinate is converted for a check.
Set the absolute values of the X axis coordinates of the barrier points as shown below:
P1 >= P2 >= P3, P4 >= P5 >= P6
(However, this need not apply to the Z axis coordinates.)

15.3.5 Interlock

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

The machine movement will decelerate and stop as soon as the interlock signal, serving as the external input, is turned ON. When the interlock signal is turned OFF, the machine starts moving again.

- In the manual mode, only that axis for which the interlock signal is input will stop.
- In the automatic mode, all axes will stop when the interlock signal is input to even one axis which coincides with the moving axis.
- Block start interlock

  While the "Block start interlock" signal (*BSL) is OFF (valid), the execution of the next block during automatic operation will not be started. The block whose execution has already commenced is executed until its end. Automatic operation is not suspended. The commands in the next block are placed on standby, and their execution is started as soon as the signal is turned ON.

  Note

  (1) This signal is valid for all blocks including internal operation blocks such as fixed cycles.
  (2) This signal (*BSL) is set ON (invalid) when the power is turned ON. If it is not used, there is no need to make a program with the PLC.
- Cutting start interlock

  While the cutting start interlock signal (*CSL) is OFF (valid), the execution of all movement command blocks except positioning during automatic operation will not be started. The block whose execution has already commenced is executed until its end. Automatic operation is not suspended. The commands in the next block are placed on standby, and their execution is started as soon as the signal is turned ON.

  Note

  (1) The signal is valid for all blocks including internal operation block such as fixed cycles.
  (2) This signal (*CSL) is set ON (invalid) when the power is turned ON. If it is not used, there is no need to make a program with the PLC.

15.3.6 External Deceleration

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

This function reduces the feed rate to the deceleration speed set by the parameter when the external deceleration input signal, which is the external input from the user PLC, has been set to ON. External deceleration input signals are provided for each axis and for each movement direction ("+" and "-"), and a signal is valid when the signal in the direction coinciding with the direction of the current movement has been input. When an axis is to be returned in the opposite direction, its speed is returned immediately to the regular speed assigned by the command.
When non-interpolation positioning is performed during manual operation or automatic operation, only the axis for which the signal that coincides with the direction of the current movement has been input will decelerate. However, with interpolation during automatic operation, the feed rate of the axis will be reduced to the deceleration rate if there is even one axis for which the signal that coincides with the direction of current movement has been input.

The external deceleration input signal can be canceled using a parameter for the cutting feed only.

15.3.7 Interference Check III

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

By checking the relative relation between interfering objects, the interference can be prevented from occurring.
One interfering object is defined by two or more solids.
The following shows the maximum numbers of solid and interfering object definitions and the maximum number of solids which can be used for one interfering object definition.

Solids/interfering objects	Number of pieces (M800VW/M800VS)	Number of pieces (M80VW/M80V)
Number of solid definitions	256 pieces	128 pieces
Number of interfering object definitions	128 pieces	64 pieces
Number of solids which can be used for one interfering object	16 pieces	6 pieces

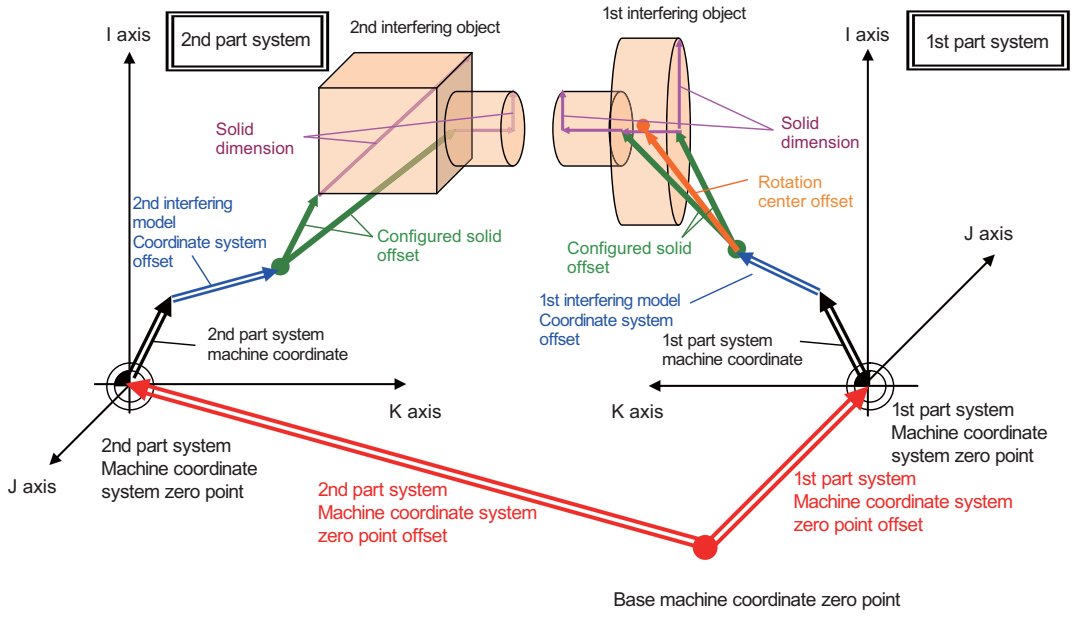
The following shows the upper limits of the number of interfering objects which can be checked and the total of solids which can be used for the interfering object definition.

Solids/interfering objects to be checked	Number of pieces (M800VW/M800VS)	Number of pieces (M80VW/M80V)
Number of interfering objects which can be checked	16 pieces	12 pieces
The total of solids which can be used in the interfering object to be checked	42 pieces	36 pieces

Select interfering objects to be checked by the R register or the system variable.

For the solids, two kinds of shapes, cuboids and cylinders, can be defined.
It is also possible to define the solid rotated by an arbitrary angle for one of the I, J, and K axes.
For the interfering object, up to 2 axes can be defined as rotary axes.
You are able to select more than one interfering object of 1-axis rotation, or one interfering object of 2-axis rotation, within interfering objects to be checked.
You are not able to select the interfering object of 1-axis rotation and the interfering object of 2-axis rotation at the same time.
2 kinds of areas can be defined for an interference area defined as the interfering object: an area where interference must be avoided surely (interference alarm area) and an area where interference should be mitigated (interference warning area)

[Schematic view of interfering object definition]

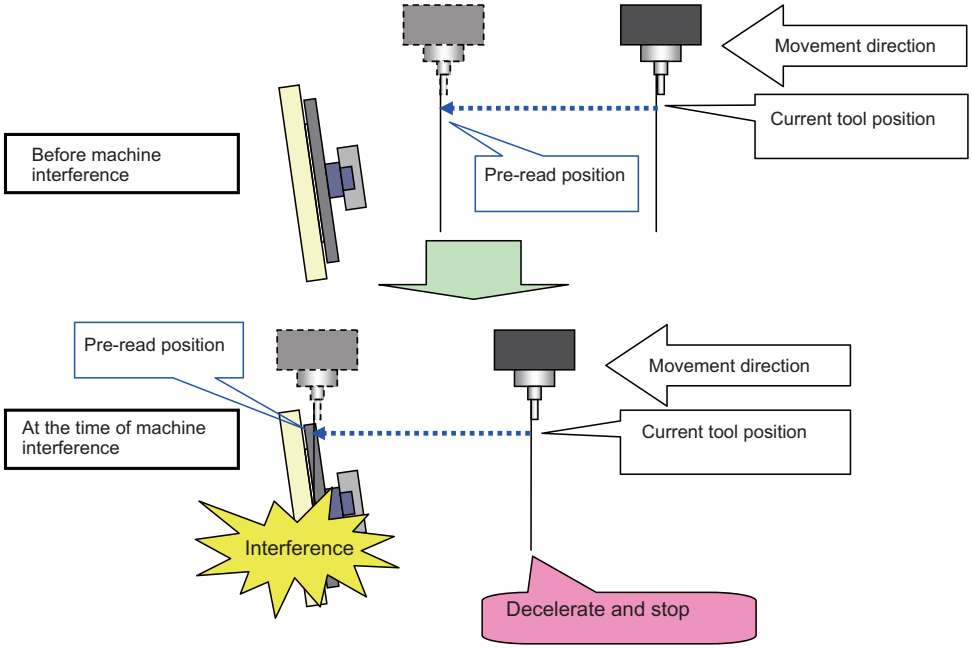


15.3.8 3D Machine Interference Check

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	—	—	○	—	—
L	—	—	—	—	—	—	—

The CNC pre-reads the position where the machine is to be moved, during automatic operation, and checks the interference by the 3D model (machine, tool and jig) registered in advance. When an interference is predicted, an alarm will be displayed immediately and all the axes will be decelerated to stop.

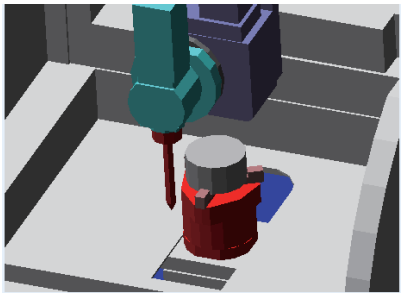
[The actual positional relation between the tool position and the interference check position]



Also, since the 3D model is displayed on the 3D monitor screen in real time, the actual positional relation of the machine can be figured out. When the interference occurs, the interference part in the 3D model will be shown in yellow or red.
The interference check is performed between the following:

- machine and machine
- machine and tool
- machine and jig
- tool and jig

The interference check is not performed between the machine and the workpiece. This function is dedicated to the machining center in single part system.



Note

(1) The software sold separately is required to create the 3D model.

15.3.9 Door Interlock

15.3.9.1 Door Interlock I

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

■ Outline

Under the CE marking scheme of the European safety standards (machine directive), the opening of any protection doors while a machine is actually moving is prohibited.
When the "Door open" signal is input from the PLC, this function first decelerates and stops all the control axes, establishes the ready OFF status, and then shuts off the drive power inside the servo drive unit so that the motors are no longer driven.
When the "Door open" signal has been input during automatic operation, the suspended machining can be resumed by first closing the door concerned and then initiating cycle start again.

■ Description of operation

[When a door is open]

The NC system operates as follows when the "Door open" signal is input:

- It stops operations.
  - When automatic operation was underway
    The machine is set to the feed hold mode, and all the axes decelerate and stop.
    The spindle also stops.
  - When manual operation was underway
    All the axes decelerate and stop immediately.
    The spindle also stops.
- The complete standby status is established.
- After all the NC axes and the spindle have stopped, the ready OFF status is established.
- The "Door open enable" signal is output.
  Release the door lock using this signals at the PLC.

[When a door is closed]

After the PLC has confirmed that the door has been closed and locked, the NC system operates as follows when the "Door open" signal is set to OFF.

- All the axes are set to ready ON.
- The "Door open enable" signal is set to OFF.

[Resuming operation]

- When automatic operation was underway
  Press the cycle start button.
  Operation now resumes from the block in which machining was suspended when the "Door open" signal was input.
- When manual operation was underway
  Axis movement is commenced when the axis movement signals are input again.
- Spindle rotation
  Restore the spindle rotation by inputting the forward rotation or reverse rotation signal again: this can be done either by operations performed by the operator or by using the user PLC.
- Chopping operation
  Input the "Chopping" signal again after returning the chopping axis to the basic point.
  If the axis is not returned to the basic point, chopping operation will be resumed from the position where the "Chopping" signal is input again.
- When spindle position control (Spindle/C axis control) is enabled.
  If the switching method of the spindle/C axis is PLC signal method and the "Door open" signal is input in the C axis mode, the mode shifts to the spindle mode. Operations when the "Door open" signal is set to OFF differ depending on the parameter setting value.

15.3.9.2 Door Interlock II

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

■ Outline

Under the CE marking scheme of the European safety standards (machine directive), the opening of any protection doors while a machine is actually moving is prohibited.
When the "Door open" signal is input from the PLC, this function first decelerates and stops all the control axes, establishes the ready OFF status, and then shuts off the drive power inside the servo amplifiers so that the motors are no longer driven.
With the door interlock function established by the "Door open II" signal, cycle start can be enabled even when the "Door open" signal has been input. However, the axes will be set to the interlock status.

■ Description of operation

[When a door is open]

The NC system operates as follows when the "Door open II" signal is input:

- It stops operations.
  All the axes decelerate and stop.
  The spindle also stops.
- The complete standby status is established.
- After all the NC axes and the spindle have stopped, the ready OFF status is established.
  However, the "Servo ready completion" signal (SA) is not set to OFF.

[When a door is closed]

After the PLC has confirmed that the door has been closed and locked, the NC system operates as follows when the "Door open" signal is set to OFF.

- All the axes are set to ready ON.
- The "Door open enable" signal is set to OFF.

[Resuming operation]

- When automatic operation was underway
  The "Door open" signal is set to OFF, and after the ready ON status has been established for all the axes, operation is resumed.
- When manual operation was underway
  Axis movement is commenced when the axis movement signals are input again.
- Spindle rotation
  Restore the spindle rotation by inputting the forward rotation or reverse rotation signal again: this can be done either by operations performed by the operator or by using the user PLC.

  Note

  (a) Concerning the handling of an analog spindle
  The signals described in this section are valid in a system with bus connections for the NC control unit and drive units. When an analog spindle is connected, the NC system cannot verify that the spindle has come to a complete stop. This means that the door should be opened after the PLC has verified that the spindle has come to a complete stop. Since the spindle may resume its rotation immediately after the door has been closed, set the forward and reverse rotation signals to OFF when opening the door so as to ensure safety.
- Chopping operation
  The "Door open" signal is turned OFF, and after the ready ON status has been established, operation is resumed. The basic point when resuming the operation is the same as where the "Chopping" signal is turned ON.
- Case in which the spindle/C axis exists
  If the switching method of the Spindle/C axis is PLC signal method and the "Door open II" signal is input in the C axis mode, the mode shifts to the spindle mode. Operations when the "Door open II" signal is set to OFF differ depending on the parameter setting value.

Note

(a) When emergency stop occurs with the "Spindle forward run start" signal or "Spindle reverse run start" signal ON, the spindle rotation does not restart even if the emergency stop is cancelled. Note, however, that if you turn ON the "Door open II" signal during emergency stop (or if emergency stop occurs with the "Door open II" signal ON), and then you cancel the emergency stop before turning OFF the said signal, the spindle rotation restarts. (When the "Door open II" signal is turned OFF first, the rotation does not restart.)

[Differences from door interlock I]

- The method used to stop the machine during automatic operation is the same as with the axis interlock function.
- The "Servo ready completion" signal (SA) is not set to OFF.
- Cycle start is valid during door interlock. However, the interlock takes effect for the axis movements.
- When the door interlock function ("Door open" signal ON) is initiated during axis movement, the axes decelerate and stop.
- When the door interlock function ("Door open" signal) is set to OFF, the axis movement resumes.
- When the door interlock function ("Door open" signal) is set to ON and OFF in the chopping operation, the chopping operation resumes.

15.3.10 Parameter Lock

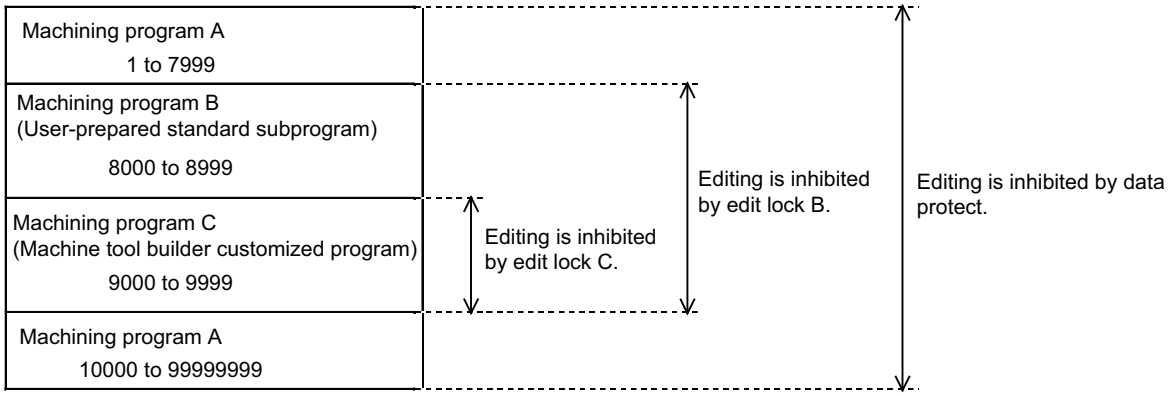
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

This function is used to prohibit changing the machine parameter.

15.3.11 Program Protection (Edit Lock B, C)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

The edit lock function B or C inhibits machining program B or C (group with machining program numbers) from being edited or erased when these programs require to be protected.



15.3.12 Program Display Lock

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

This function allows the display of only a target program (label address 9000) to be invalidated for the program display in the monitor screen, etc.
The operation search of a target program can also be invalidated.
The validity of the display is selected with the parameters. The setting will be handled as follows according to the value.

0: Display and search are possible.
1: Display of the program details is prohibited.
2: Display and operation search of the program details are prohibited.

15.3.13 Data Protection by User's Level

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

Up to 8 levels of access permission helps to prevent you from dispatching defective works.
On the protection setting screen, the current operation level can be displayed and changed, and the password for the operation level and the protection level for each data can be changed.

15.3.15 Vertical Axis Pull-Up

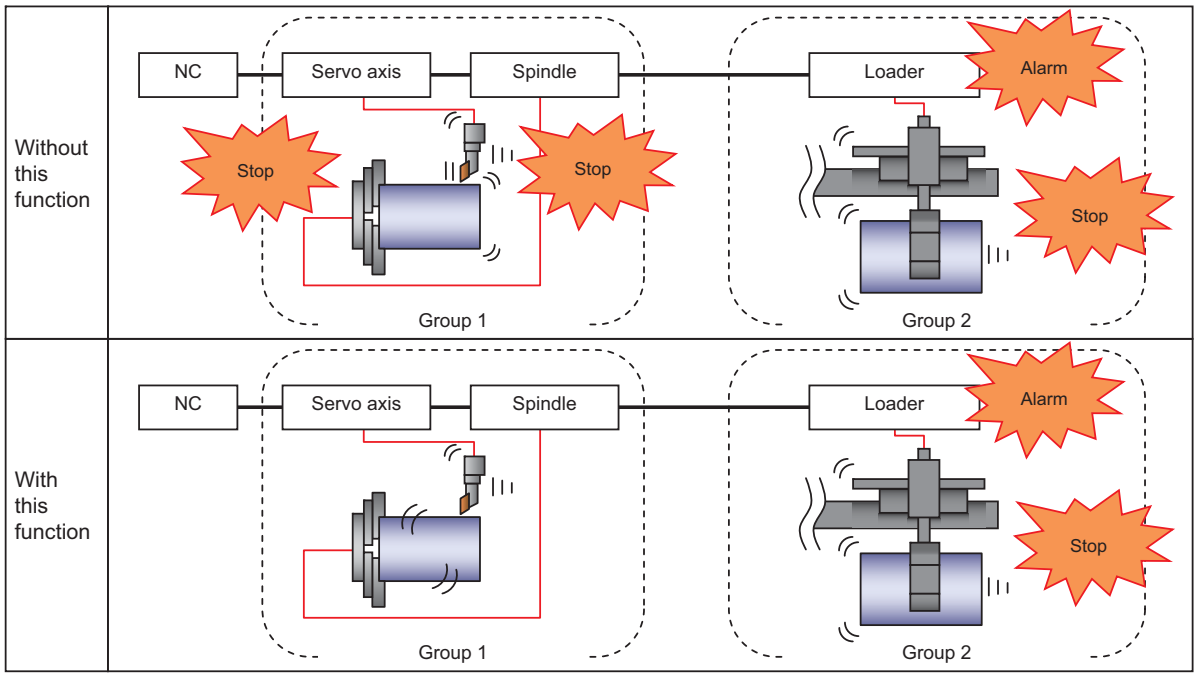
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

When emergency stop or power interruption occurs, motor brake and vertical axis drop prevention function is supposed to work to prevent vertical axis (such as Z axis of vertical machining center) from dropping. However, there might be a case that the vertical axis drops by several μm because of decrepitude of motor brake.
Therefore, emergency stop or the power interruption during finish machining at low speed might cause damage to the cutting tool in operation.
Vertical axis pull-up function prevents the tool from breakage, through pulling up the cutting tool during emergency stop or instantaneous power interruption at low cutting speed.

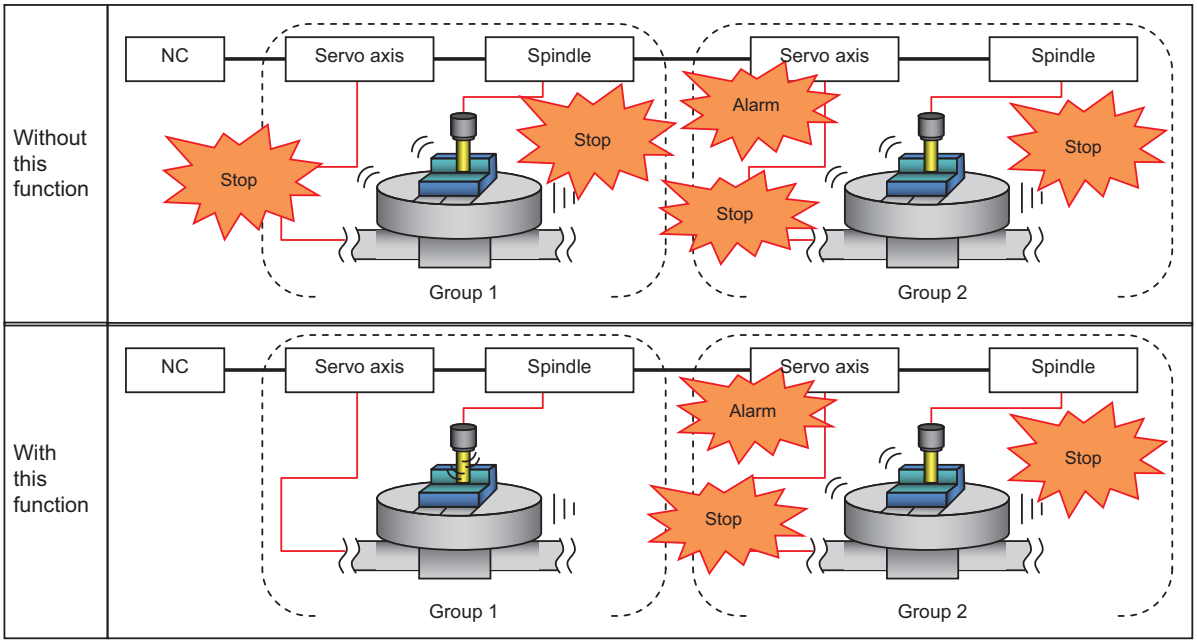
15.3.16 Machine Group-Based Alarm Stop

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

When an alarm occurs for an axis, this function performs an alarm stop only for the axes in a machine group to which the axis belongs. When this function is used in a lathe system, even though an alarm occurs for the loader axis, the operation can be continued if the servo axis and the spindle have been set in a different machine group from the loader axis as illustrated below. Without this function, the alarm stop is performed for all the axes when an alarm occurs.



Likewise, when this function is used in a machining center system, the axes belonging to a different machine group from the axis for which an alarm has occurred can continue the operation as illustrated below. Without this function, the alarm stop is performed for all the axes when an alarm occurs.



Note

(1) The machining group number can be set for each servo axis and spindle in this function, and incorrect setting may lead to accidents. Set the machine group number with enough care.

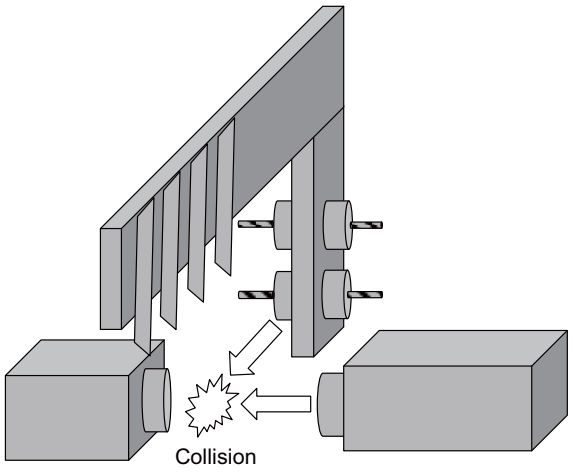
15.3.17 Interference Check Between Part Systems

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	O	O	O	O	O	O	—

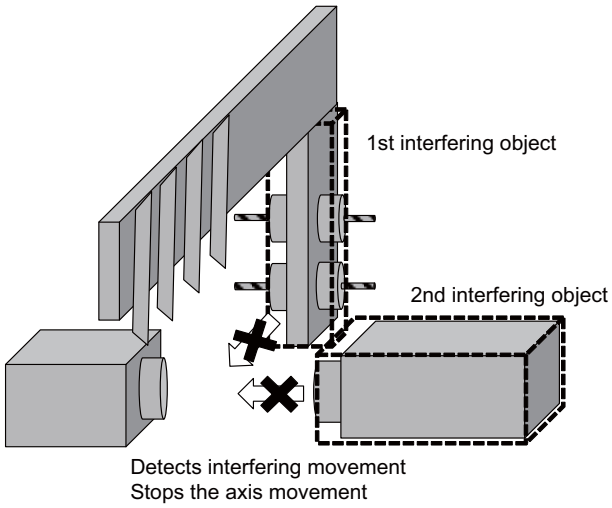
Maximum 4 part systems

This function checks the relative position of up to six cuboids (referred to as interfering objects) all the time, and if a command which causes the interfering objects to collide with each other is issued, the function stops the axis movement to prevent the interference in advance.
Such interference can be prevented by covering the tool post, etc. with cuboids. Because the check is performed for axis movement of all part systems, interference between tool posts which are positioned in difference part systems can be prevented.
This is valid for the L-system configured with two or more part systems.

<Interference check between part systems OFF>



<Interference check between part systems ON>



15.3.18 Spindle Protection

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

By setting the temperature to start/cancel the spindle protection, acceleration/deceleration time constant of the spindle is changed automatically when the spindle is overloaded and the operation is continued while suppressing overheating of the spindle.
This is effective in protecting the spindle when the spindle is overloaded by repeated acceleration/deceleration of the spindle.
Protection state of the spindle is output to the "Spindle protection is active" signal (for each spindle). When acceleration/deceleration time constant of the spindle has been changed to protect the spindle, this signal can be used to notify users that the spindle is protected.

The equivalent load ratio of spindle motor is determined by calculating the average output of spindle motor from the start of cycle operation to the next cycle operation and showing the average output as a percentage of the continuous rated output. The ratio is output to the ZR register and the drive monitor (item name: Duty value).

The equivalent load ratio and temperature of spindle motor can be output to a log file.
The log file can be used to analyze the operating environment of spindle motor and to detect a decrease in cooling capacity caused by dirt or clogging of spindle motor cooling fan.

While the same machining program is operated continuously, if the equivalent load ratio of last operation exceeds the threshold at the start of cycle operation, a warning is displayed.

15.3.19 Real-Time 3D Machine Interference Check

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	—	—	—	—	—
L	—	—	—	—	—	—	—

The CNC pre-reads the position where the machine is to be moved, during automatic operation, and checks the interference by the 3D model (machine, tool and jig) registered in advance. When an interference is predicted, an alarm will be displayed immediately and all the axes will be decelerated to stop. Since the 3D model is displayed on the 3D monitor screen in real time, the actual positional relation of the machine can be figured out.

15.4 Maintenance and Troubleshooting

15.4.1 Operation History

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

This is a maintenance function which is useful for tracing down the history and NC operation information and analyzing trouble, etc. This information is saved in the history data file, and can be displayed on the screen and output to a file.
The machine tool builder password is required to use this function.

The following of history data files are used.

All history	The key history, touchscreen history, alarm/warning history, PLC signal history, tool offset change history, workpiece offset change history, AC input power OFF history, parameter change history (*1), and common variable change history (*1) are recorded in date/time order.
Key history	Only the key history is recorded in date/time order.
Key touchscreen history	Only the touchscreen history is recorded in date/time order.
Detailed alarm history information	Separately from the existing alarm history, the alarm information, modal information and coordinate information at the time of the alarm occurrence are recorded in date/time order.

The data above can be viewed on the edit screen.
(*1) This information can be viewed on the setting change history screen. Note that it is not output to the history data files.

The data sorting order of the all history data files can be changed by the parameter.
- In chronologically ascending order for each log type
- In chronologically ascending order for all the log types

Note

(1) If the date and time are the same, the history will be displayed in the following order.
Key history
Touchscreen history
Alarm history
PLC signal history
Tool offset change history
Workpiece offset change history
AC input power OFF history
Parameter change history
Common variable change history

15.4.2 Data Sampling

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

The NC internal data (speed output from NC to drive unit, feedback data from the drive unit, etc.) can be sampled and the data can be output as text data.
The type of sampling NC internal data and sampling conditions are input on the screen as the sampling parameter. The sampling parameter can be input/output with the sampling data as the header information of the sampling data.
The parameter can be used to set whether the sampling starts when the power is turned ON.
Sampling data can be displayed in waveform by channel.

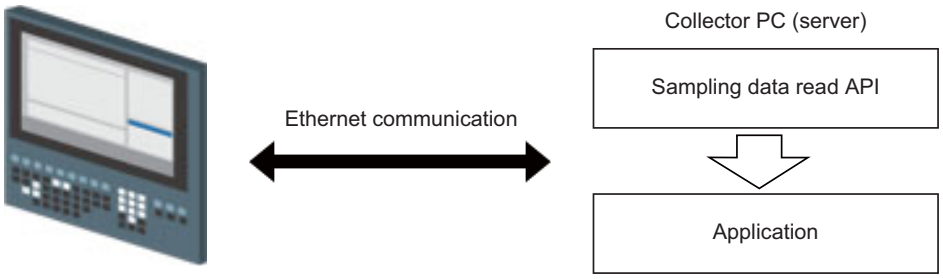
Item	Specifications	
Sampling cycle	1.776 ms x setting value	
Number of sampled axes	NC axis: The maximum number of axes usable for specifications Spindle: The maximum number of spindles usable for specifications	
Number of sampled channels	1 to 16 points 1 to 8 points for waveform display	
Sampling data size	M800V	Maximum 2,620,416 points
	M80V	Maximum 655,360 points
	<Note> • This is the entire data size. The data size per channel will decrease when the number of sampled channels increases.	

15.4.2.1 Real-time Data Sampling

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

This function saves sampling data obtained in NC, in a PC (collector PC) on network in real time (*1).
The sampling data read API obtains arbitrary sampling data and passes it to the application (*2), which enables machining diagnosis, operation monitoring, etc.
Up to four sets of sampling control (condition setting, start, end, read, etc.) are available.
Up to 32 channels are available in total for the 4 sets. There is no upper limit of the number of channels per set.
(*1) NC transmits the sampling data to the collector PC within 56.8 ms. In a system with 5 or more part systems or with 17 or more axes, NC transmits the data within 113.7 ms that is twice.
(*2) This function does not include the application.

Ethernet communication is performed between NC and collector PC (server).
The sampling data is obtained through the sampling data read API.



15.4.3 NC Data Backup

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

The NC data back up function backs up the NC parameters, etc., on a built-in disk of display (HD) unit or SD card.
The data can also be restored.

15.4.4 Servo Tuning Support

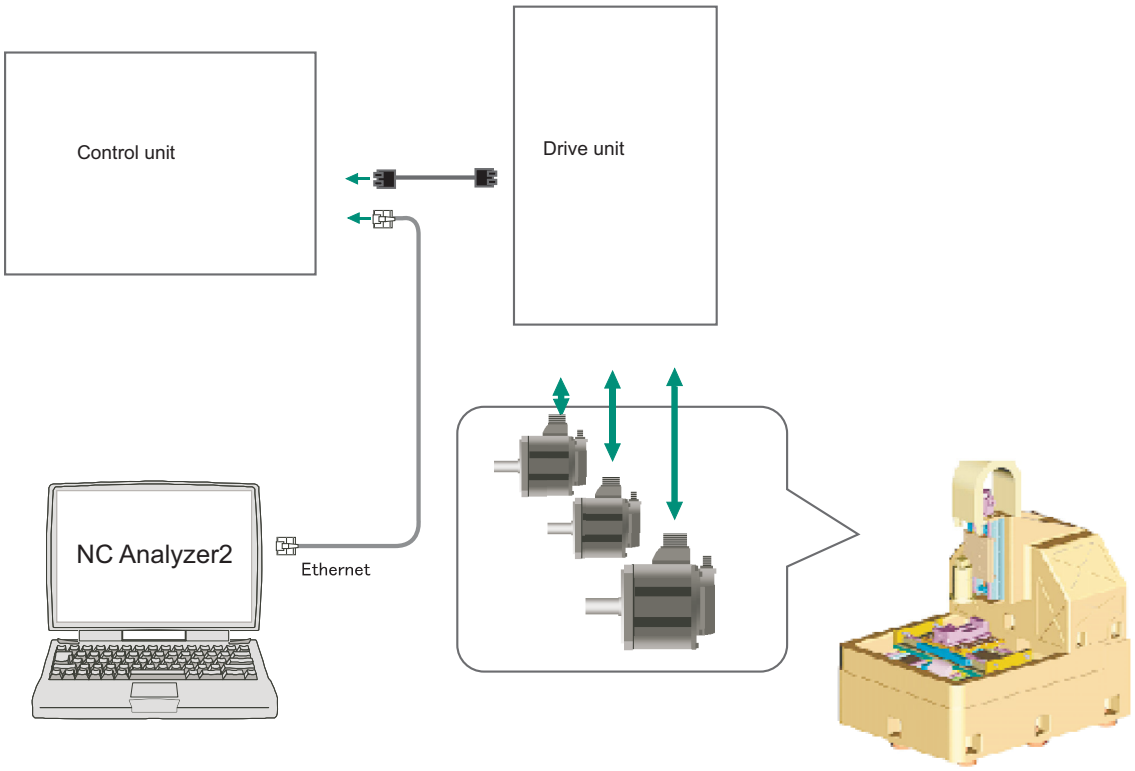
15.4.4.1 NC Analyzer2

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

Note

(1) The software sold separately is necessary.

Servo parameters can be automatically adjusted by measuring/analyzing the machine characteristics. The measurement/analysis is conducted by activating the motor using machining programs for adjustment or vibration signals. Various data can be sampled.



■ Function

Adjustment wizard	Description
Velocity loop gain adjustment	Automatically adjusts velocity loop gain and resonance filters.
Lost motion adjustment	Adjusts the lost motion automatically for the delay in response caused when the direction of the machine rotates is reversed.

Measurement and adjustment	Description
Frequency response measurement	Measures the frequency response of speed loop for the designated axis. The result will be output as frequency response waveform.
Reciprocation acceleration/deceleration measurement	Measures the reciprocation acceleration/deceleration for the designated axis. The result will be output as time-series waveform.(*1)
Roundness measurement	Measures the roundness for the designated axis. The result will be output as roundness waveform.(*1)
Synch tap error measurement	Measures the synchronous tap error for the designated axis. The result will be output as time-series waveform.(*1)
Spindle acceleration/deceleration measurement	Measures the spindle acceleration/deceleration for the designated axis. The result will be output as time-series waveform.(*1)
Spindle orientation measurement	Measures the spindle orientation for the designated axis. The result will be output as time-series waveform. (*1)
Spindle C-axis measurement	Executes the reciprocation acceleration/deceleration measurement of spindle C axis with specified axis. The result will be output as time-series waveform. (*1)
Spindle synchronization measurement	Executes an arbitrary machining program with specified axis, and measures spindle synchronization by using that data. The result will be output as time-series waveform.
PLC axis acceleration/deceleration measurement	Executes an arbitrary machining program with specified axis, and measures PLC axis acceleration/deceleration by using that data. The result will be output as time-series waveform.
Arbitrary path measurement (*2)	Executes an arbitrary machining program with specified two axes. The result will be output as arbitrary path measurement waveform. (*1)

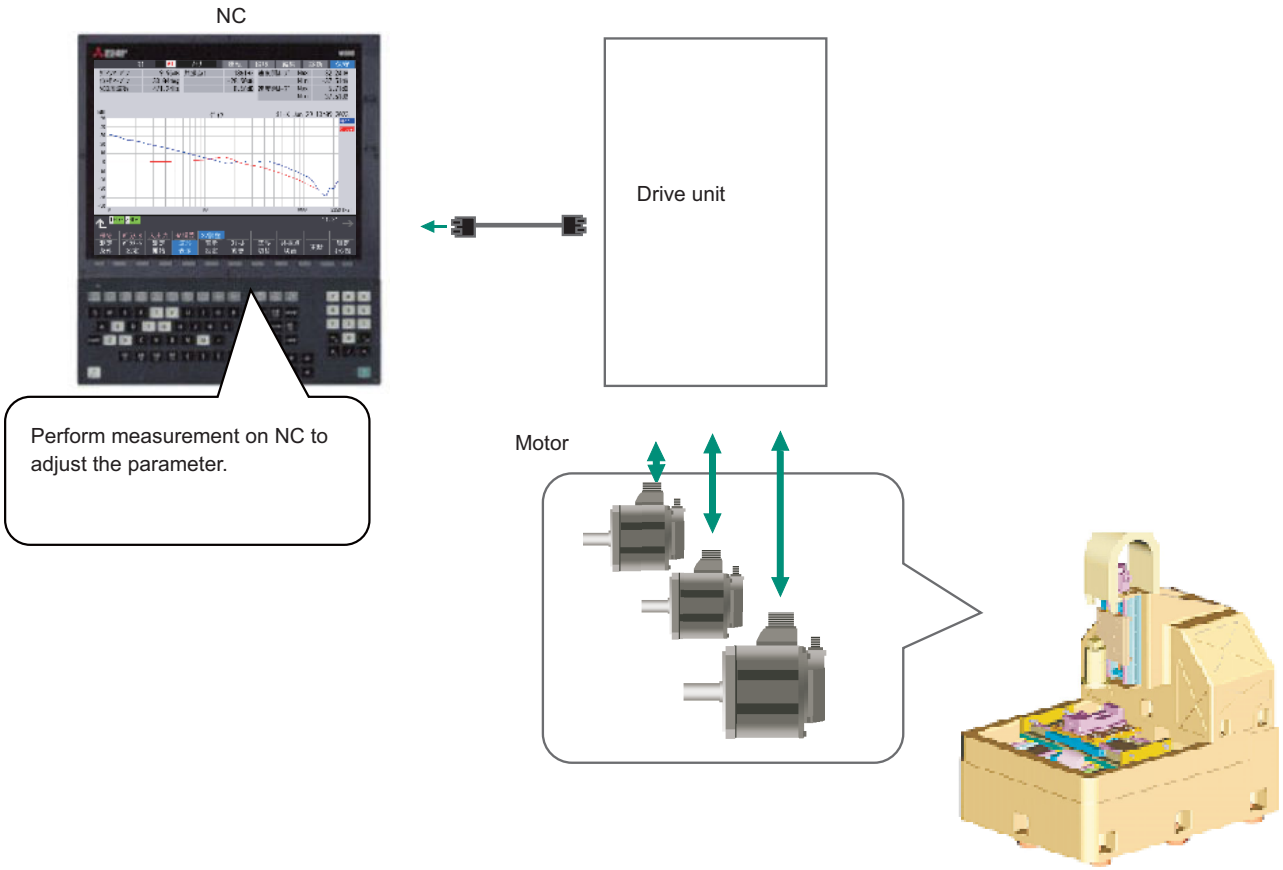
(*1) For the measurement, create a program by the program creation function or use an arbitrary machining program.

(*2) Two or more NC axes in the same system are needed.

15.4.4.3 On-board Servo Tuning

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

The on-board servo tuning is a function to measure characteristics of the servo control system and support adjustment with parameters. The measurement is performed by vibrating the motor with vibration signals and measuring/analyzing the feedback.



■ Function

Function	Details
Frequency response measurement	Measures the frequency response of speed loop for the designated axis. The result will be output as frequency response waveform.
Reciprocation acceleration/deceleration measurement	Measures the reciprocation acceleration/deceleration for the designated axis. The result will be output as time-series waveform. (*1)
Roundness measurement	Measures the roundness for the designated axis. The result will be output as roundness waveform. (*1)
Velocity loop gain adjustment	Automatically adjusts velocity loop gain and resonance filters.

(*1) For the measurement, use a measurement program created by the program creation function of this function or an arbitrary measurement program.

15.4.5 Automatic Backup

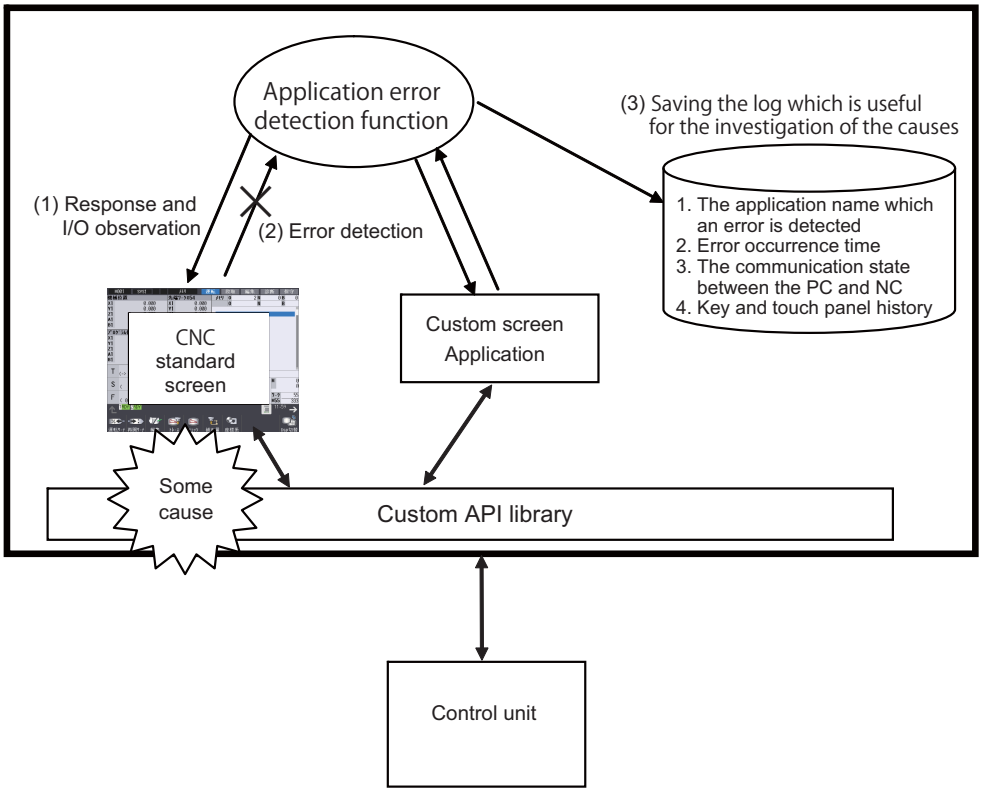
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

With this function, system data, ladder program and custom software can be automatically backed up in case of system failure. Up to the three most recent backup data can be stored.
The backup execution timing can be set with parameters.

15.4.8 Application Error Detection

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	—	—	O	—	—
L	O	O	—	—	O	—	—

Application error detection function observes applications such as the CNC standard screen or custom screen. When an error such as screen lock is detected, this function saves information and data in the log to investigate the causes easily.

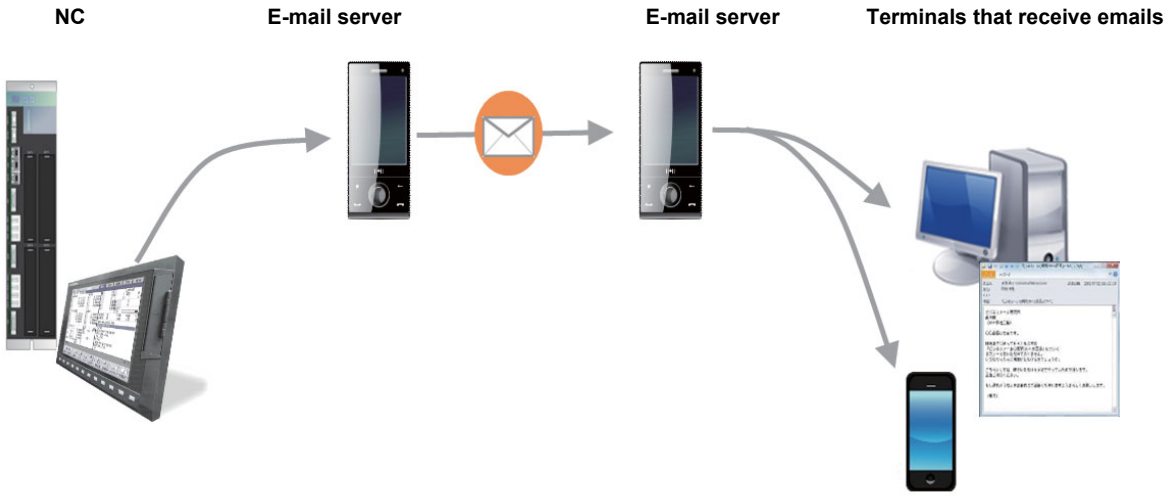


- Response from the application and a change of I/O is observed on a periodical basis.
- When an application with no response and no change of I/O for a certain period is detected, it is determined that there is an error.
- When an error is detected, the log which is useful for the investigation of the causes will be saved.

15.4.9 Email Notification to Operator

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

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. With this function, the NC can send emails to PCs and mobile terminals away from machines. You are able to know machining conditions (such as machining completion, stop and failure) even if you are in remote places.



15.4.10 NC Configurator2

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

NC Configurator2 achieves the following functions for the multiple NCs connected to the network, or a file in the SD card or the local disk. Data from up to 8 machines can be handled at a time.

NC Data	Transmit, display, edit and control the data related to the NC machine and parameters for NC control module side, and transmit, display the machining programs, tools and work.
Function parameter	Support of parameter settings for individual NC function for the users.
RS232C communication function:	Offer various functions using the tape mode and serial communication.

Please contact us to purchase a full function version. A limited function version is also available free of charge.

15.4.11 Diagnosis Data Output

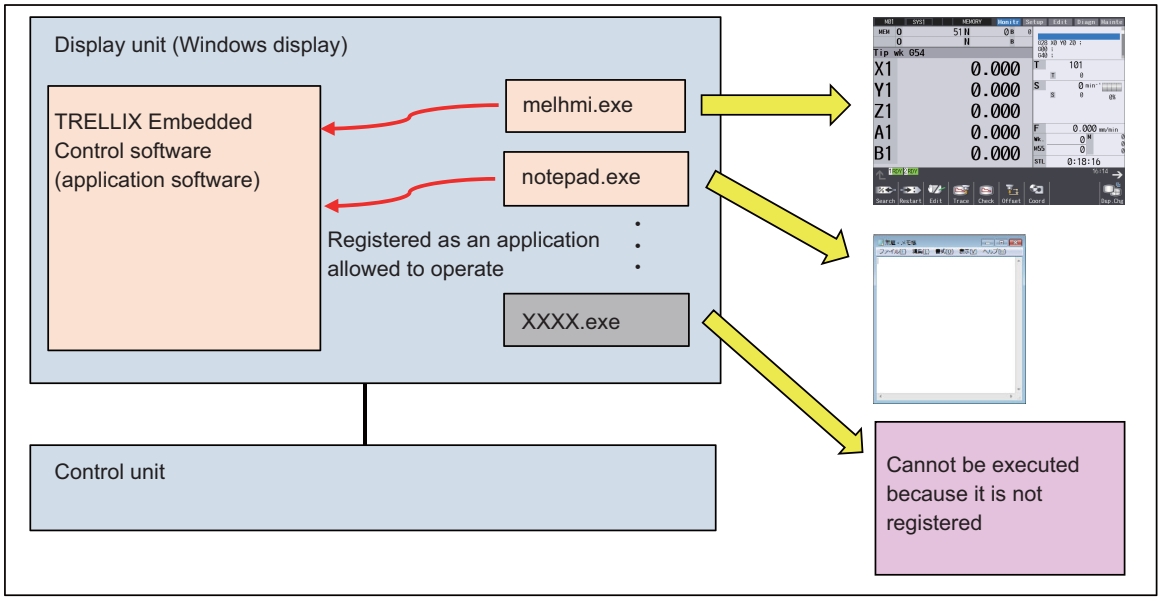
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

With this function, the information indicating the replacement cycle of the service parts used in NC, drive or motor can be output to the ZR registers. The information, which is output to ZR registers, can be used for preventive maintenance, including lifetime prediction.

15.4.13 Security Feature for Windows Display

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	—	—	—	—	—
L	Δ	Δ	—	—	—	—	—

This function only activates applications that have been allowed to work when the TRELLIX Embedded Control software is installed on the display unit. This software restricts the operations of the application. To allow the application to work, the TRELLIX Embedded Control software must be registered in the list of applications which is allowed to start.



15.4.14 Field Network Diagnostics

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	□	□	□	□	□
L	□	□	□	□	□	□	□

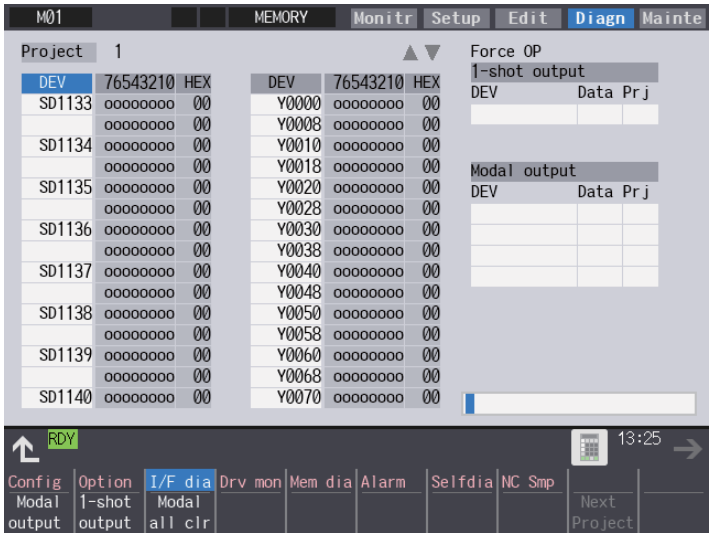
This function displays the statistical information of packet transmission and reception (transmission/reception frequency, number of errors, etc.), which is necessary when the communication status of the function expansion unit (the field network expansion unit) is diagnosed.
The function is useful to figure out the causes, for example, in the following cases.

- Communication is disconnected
- Communication is not carried out properly

The function expansion units in the table below are available with this function.

Unit No.	Protocol
FCU8-EX564	CC-Link IE Field
FCU8-EX565	EtherNet/IP
FCU8-EX568	FL-net

The statistical information of communication packets can be checked on the I/F diagnostic screen of NC.

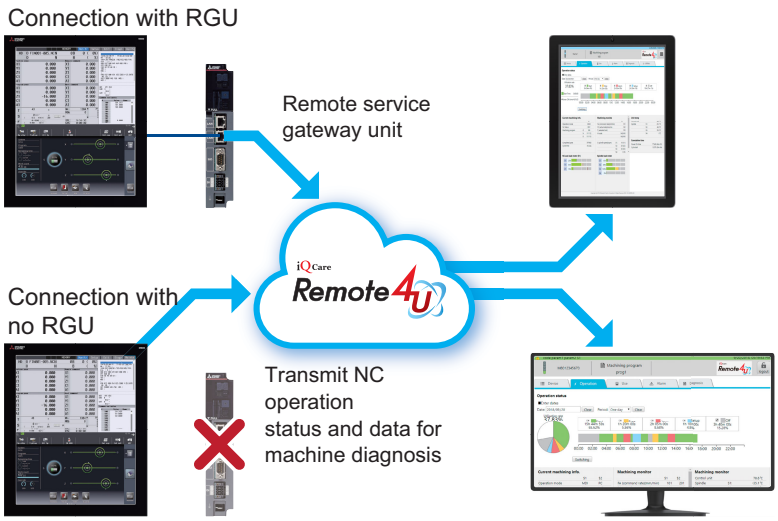


15.4.15 Remote service iQ Care Remote4U

This function is a remote service to support remote maintenance of machine tools using IoT technology. Operation monitoring of machine tools and remote diagnostics of CNC are available.

Note

(1) The license agreement is required.



15.4.15.1 Connection with RGU

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	□	□	□	□	□
L	□	□	□	□	□	□	□

A remote service gateway unit is required separately.

Connection with RGU connects the remote service via a remote service gateway unit (RGU).

15.4.15.2 Connection with no RGU

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

Connection with no RGU uses only CNC to connect the remote service without using the remote service gateway unit (RGU).

Note

(1) If the data needs to be retrieved at a 1-second interval, use the remote service gateway unit. (When connection with no RGU is used, the data is retrieved at a 30-second interval.)

(2) Connection with no RGU can be used in the configuration where there are 3 or less part systems and the total number of axes of the NC is 12 or less. If there are more part systems or axes, use the remote service gateway unit.

15.4.16 Brake Wear Diagnostics

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

This function diagnoses the wear state of a brake installed on each axis of the machine tool.
When a machine runs for a long time, the brake body and the fitting hub are gradually worn. The degree of brake wear is diagnosed by measuring the amounts of rattling and slipping which increase as the wear progresses.

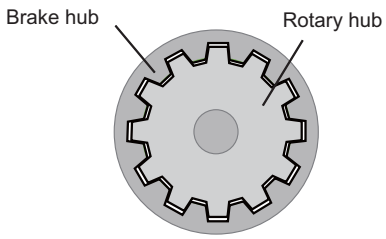
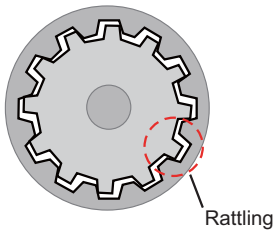
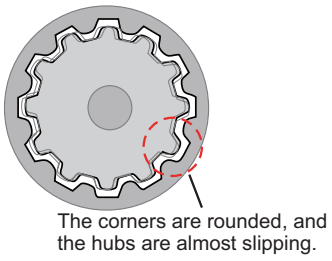
The three levels of Normal, Caution and Warning are output as the diagnosis result.

Normal state	The rattling of the brake is within the normal range.
Caution state	PLC is informed that the brake is in the caution state. Automatic operation is possible.
Warning state	PLC is informed that the brake is in the warning state. Automatic operation is not possible.

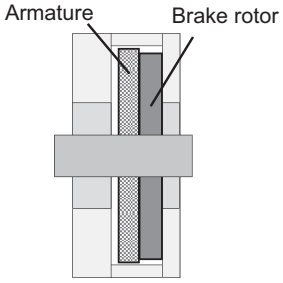
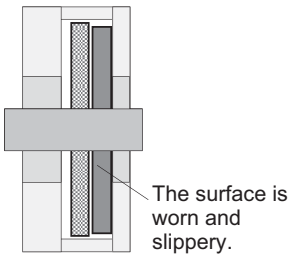
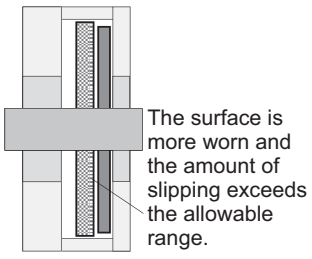
By executing this function periodically, the amounts of rattling and slipping caused by the brake wear can be maintained within the proper range.
All the diagnosis results are output to PLC. By combination with PLC program, it is possible to inform users of the time to change the brake before it is broken.

This function has a timer function. After a certain period of time has passed since the diagnosis, a signal prompting the next diagnosis is output to PLC. This makes periodic brake wear diagnosis by combination with PLC program.

[Wear state and diagnosis result of brake fitting hub]

Normal state	Caution state	Warning state
		

[Wear state and diagnosis result of brake rotor and armature]

Normal state	Caution state	Warning state
		

15.4.17 NC MachiningAID

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

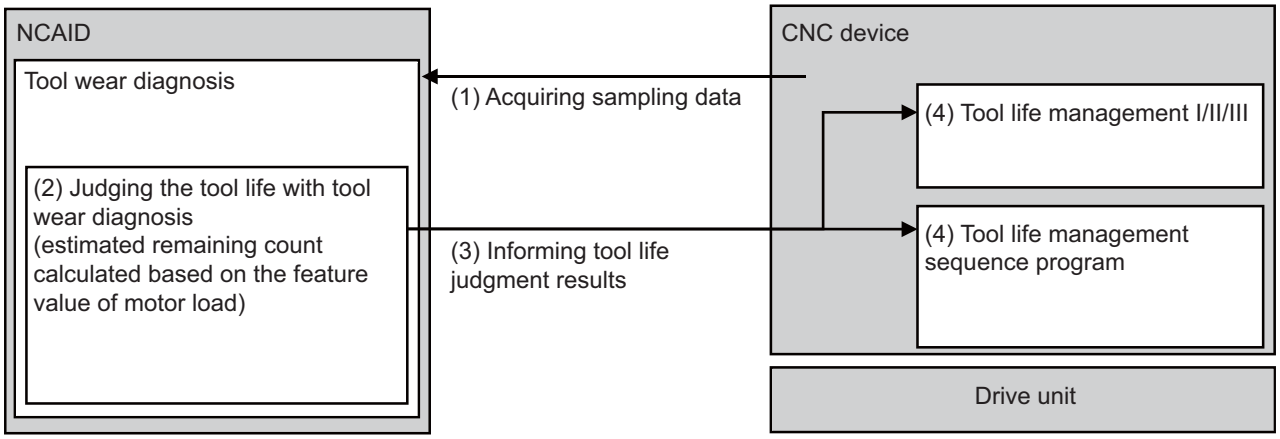
Note

(1) The software sold separately is necessary.

NC MachiningAID (NCAID) is a product to enable stable operation and machining cost reduction of machine tools (such as machining centers, milling machines, and tapping centers). It can be applied to machining diagnosis targeting abnormal machining and cutting tool wear.
By automatically collecting and analyzing data during machining on a machine tool, the operation of the cutting tool transitions from the time based maintenance (TBM) to condition based maintenance (CBM). This supports equipment error prevention, tool cost reduction, and quality defect prevention.
This is used by connecting between NC and PC equipped with NCAID through a LAN.

[Tool wear diagnosis]
This is a function to inform the NC tool life management function or sequence program of the NCAID tool life judgment results. By linking the NCAID tool wear diagnosis and the tool life management, the tool life management can be performed based on the estimated remaining count calculated from the feature value of motor load instead of the tool cutting hours or the number of cuttings.
The NCAID tool life judgment results can be informed to arbitrary R registers. NCAID can be also introduced when the tool life management is performed with sequence programs.
With the NCAID tool life judgment, tools can be used efficiently, and the costs of tool change can be reduced.

This function operates as follows.
(1) NCAID acquires sampling data during machining.
(2) The NCAID tool wear diagnosis calculates the estimated remaining count based on the machining conditions and the motor load.
(3) The NCAID tool life judgment results are informed to the tool life management or sequence program.
(4) Based on the tool life judgment results, tool change, etc. is performed with the NC tool life management function or sequence program.



15.5 Functional Safety

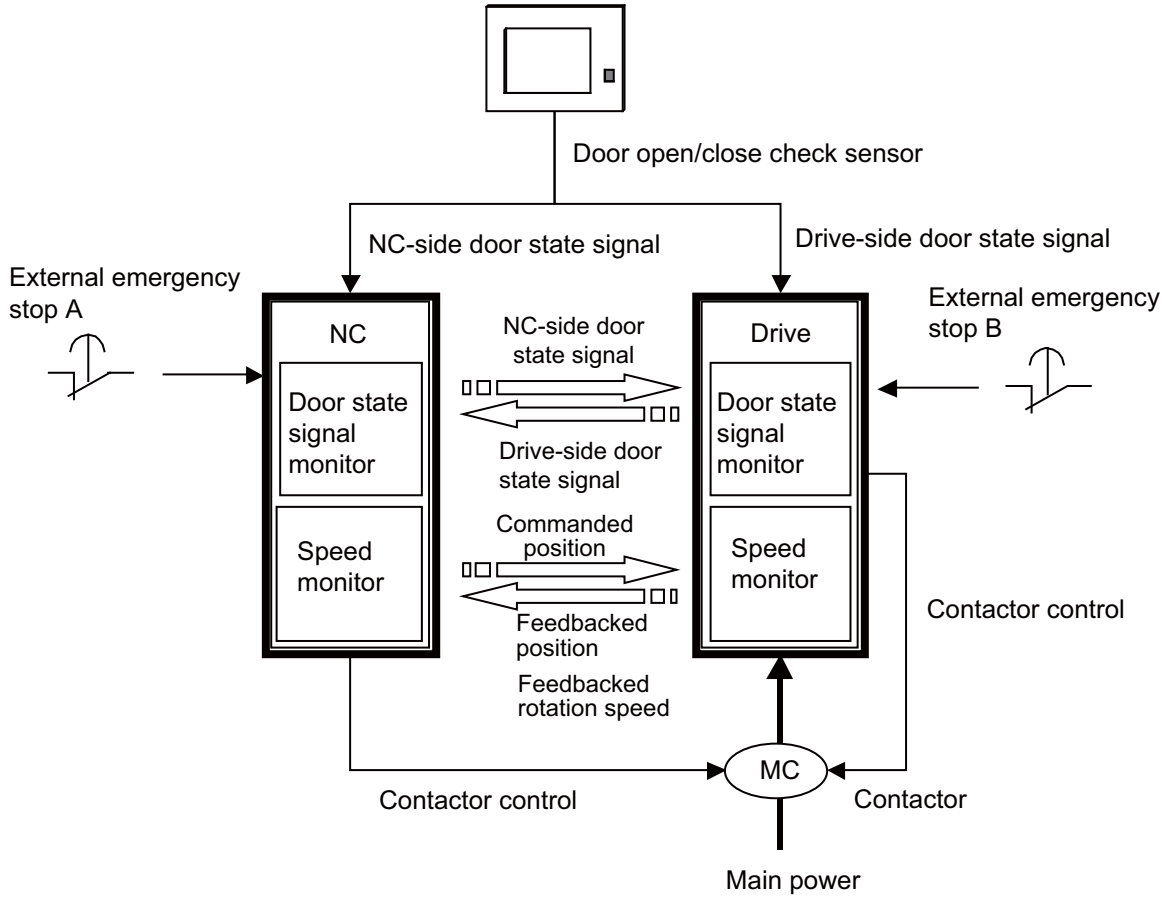
15.5.1 Safety Observation

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

This function can be available in the system consisting of the NC control unit and drive unit, and various communication cables, sensors and contactors.
The followings are available.

- Opens the door without shutting off the drive power of motor.
- Operates at safety speed or lower while the door is opened.

Function	Outline
Dual emergency stop	The NC control unit and the drive unit separately observe the input of emergency stop. In case of an emergency stop, both the NC control unit and the drive unit (power supply unit) will independently control the contactors to shut the drive power OFF.
Speed monitor	The NC control unit and the drive unit (servo/spindle drive unit) separately observe the following. When an error is detected during observation, the main power for the drive will be shut OFF. • That the command speed should not exceed the speed set by the parameter (the "safety speed"). • That the motor rotation speed should not exceed the rotation speed set by the parameter (the "safety rotation speed").
Door state signal monitor	This observes the door state signal (open or closed) detected with different circuit.



15.5.2 Smart Safety Observation

The smart safety observation function achieves smart and high safety with the following features.

- The redundant observation is enabled not only for the door signal but for various signals that require the safety.
- Eight new functions including the position observation are added.
- Signal wiring is simplified by having redundant PLC signal. (With the conventional safety observation, the signal wiring is needed for NC and drive units.)
- Redundant safety ladder eliminates the need for MTB to prepare the safety unit.
- The dual safety ladder is enabled by making only one ladder if they serve as the equivalent role. (It is also possible to make two ladders to have different roles.)
- The I/O board equipped with safety signal input terminal is available, which is mounted on the back of the operation panel. This eliminates the need to mount the safety I/O unit additionally in the operation panel.

Note

(1) Safety I/O devices (safety RIO unit, operation panel with safety I/O) are required for inputting/outputting safety signals. Of the I/O device connection channels on NC unit, the channels which input/output safety signals need to be composed only of safety I/O devices and RIO2.0 units. (RIO1.0 units cannot be connected to I/O channels which input/output safety signals.)
(2) For user safety sequence, machine tool builders need to create safety signal process logic and write it to NC unit.
(3) Of connection channels for each drive unit in NC unit, channels with axes which use "Safety function S" need to be composed only of drive units of MDS-E Series.

15.5.2.1 Safety-Related I/O Observation

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	□	□
L	Δ	Δ	Δ	Δ	○	□	□

Functional safety expansion unit is required for M80V.

Using the dual circuits for processing signals input/output to/from the machine (safety signal compare sequence) and dual execution of safety signal process logic made by users (safety PLC), if one circuit has broken down, the other circuit can detect errors, which improves the safety of signal process.

15.5.2.2 Emergency Stop Observation

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	□	□
L	Δ	Δ	Δ	Δ	○	□	□

Functional safety expansion unit is required for M80V.

Emergency stop signal is doubled and observed to see whether there is any error. When one emergency stop signal is in open state, the whole system can be set in emergency stop condition.

15.5.2.3 Drive Safety Function

15.5.2.3.1 SLS (Safely-Limited Speed)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	□	□
L	Δ	Δ	Δ	Δ	○	□	□

Functional safety expansion unit is required for M80V.

Axis speed (command speed, FB speed) is observed doubly to see whether the speed exceeds the safe speed. Four steps of safe speed and 16 steps of override can be selected. (64 ways of safe speed can be created in total.)

15.5.2.3.2 SLP (Safely-Limited Position)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	□	□
L	Δ	Δ	Δ	Δ	○	□	□

Functional safety expansion unit is required for M80V.

Axis absolute position (command position, FB position) is observed doubly to see whether the position exceeds the safe position range. Safe position range can be selected from 4 levels.

15.5.2.3.3 SOS (Safe Operating Stop)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	□	□
L	Δ	Δ	Δ	Δ	○	□	□

Functional safety expansion unit is required for M80V.

SOS is made by performing the following operations simultaneously.

- Axis stop speed (command speed, FB speed) is observed doubly whether the speed exceeds the safe stop speed.
- Axis stop position (command position, FB position) is observed doubly whether the position exceeds the safe stop position range.
- Axis stop position deviation (difference between command position and FB position) is observed doubly to make sure that the deviation does not exceed the safe stop position deviation.

15.5.2.3.4 SSM (Safe Speed Monitor)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	□	□
L	Δ	Δ	Δ	Δ	○	□	□

Functional safety expansion unit is required for M80V.

This function uses the safety signals to inform that the axis speed (command speed, FB speed) is equal to or below the safe speed. Up to 4 safe speeds can be set.

15.5.2.3.5 SBC/SBT (Safe Brake Control/Safe Brake Test)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	□	□
L	Δ	Δ	Δ	Δ	○	□	□

Functional safety expansion unit is required for M80V.

This function activates the brake connected to the motor. Because there are two circuits for activating the brakes, one circuit can activate the brakes even when the other circuit is broken down. Furthermore, Safe Brake Test (SBT) can diagnose the circuits for activating the breaks and the effectiveness of the brakes (deterioration due to abrasion, etc.).

15.5.2.3.6 SCA (Safe Cam)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	□	□
L	Δ	Δ	Δ	Δ	○	□	□

Functional safety expansion unit is required for M80V.

This function uses the safety signals to inform that the axis absolute position (command position, FB position) is within the range of safe position. Up to 16 sets (16 each for pulse side and minus side) can be set for the safe position range.

15.5.2.3.7 SS1/SS2 (Safe Stop)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	□	□
L	Δ	Δ	Δ	Δ	○	□	□

Functional safety expansion unit is required for M80V.

■ Safe stop 1 (SS1)

STO function is activated after an axis is decelerated and the speed (command speed, FB speed) becomes equal to or below the safe stop speed. STO function is also activated when the speed does not reach the safe stop speed within a specified time from the start of deceleration.

■ Safe stop 2 (SS2)

SOS function is activated after an axis is decelerated and the speed (command speed, FB speed) becomes equal to or below the safe stop speed. SOS function is also activated when the speed does not reach the safe stop speed within a specified time from the start of deceleration.

15.5.2.3.8 STO (Safe Torque Off)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	□	□
L	Δ	Δ	Δ	Δ	○	□	□

Functional safety expansion unit is required for M80V.

This function shuts OFF power supply to axes. Because there are two power shutoff circuits, one circuit can shut OFF the power supply even when the other circuit is broken down. STO also performs fault diagnosis of the shutoff circuits.

16

Drive System

16.1 Servo/Spindle

Refer to the following manuals for details on the servo and spindle system.
DRIVE SYSTEM DATA BOOK (IB-1501252 (ENG))
MDS-E/EH Series Specifications Manual (IB-1501226 (ENG))
MDS-EJ/EJH Series Specifications Manual (IB-1501232 (ENG))
MDS-EM/EMH Series Specifications Manual (IB-1501238 (ENG))

16.1.1 Feed Axis

16.1.1.1 MDS-E-Vx

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	□	□	□	□	□
L	□	□	□	□	□	□	□

16.1.1.2 MDS-EH-Vx

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	□	□	□	□	□
L	□	□	□	□	□	□	□

16.1.1.3 MDS-EJ-Vx

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	□	□	□	□	□
L	□	□	□	□	□	□	□

16.1.1.4 MDS-EJH-Vx

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	□	□	□	□	□
L	□	□	□	□	□	□	□

16.1.1.5 MDS-EM-SPVx

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	□	□	□	□	□
L	□	□	□	□	□	□	□

16.1.1.6 MDS-EMH-SPVx

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	□	□	□	□	□
L	□	□	□	□	□	□	□

16.1.2 Spindle

16.1.2.1 MDS-E-SPx

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	□	□	□	□	□
L	□	□	□	□	□	□	□

16.1.2.2 MDS-EH-SPx

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	□	□	□	□	□
L	□	□	□	□	□	□	□

16.1.2.3 MDS-EJ-SPx

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	□	□	□	□	□
L	□	□	□	□	□	□	□

16.1.2.5 MDS-EM-SPVx

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	□	□	□	□	□
L	□	□	□	□	□	□	□

16.1.2.6 MDS-EMH-SPVx

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	□	□	□	□	□
L	□	□	□	□	□	□	□

16.1.4 Power Supply

16.1.4.1 MDS-E-CV

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	□	□	□	□	□
L	□	□	□	□	□	□	□

16.1.4.2 MDS-EH-CV

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	□	□	□	□	□
L	□	□	□	□	□	□	□

16.1.4.3 MDS-EX-CVP

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	□	□	□	□	□
L	□	□	□	□	□	□	□

17

Machine Support Functions

17.1 PLC

17.1.1 Built-in PLC Processing Mode

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

An exclusive sequence program that controls the various signals between the controller and machine to realize operation applicable to each machine can be created and built in.
The sequence execution modes include high-speed processing and main processing.

■ High-speed processing

This mode provides repeated execution at constant cycles. It is used to process signals requiring high speeds.
The maximum number of program steps for high-speed processing (1 period) is 1000 steps when using basic instructions.

■ Main processing

This mode provides normal sequence processing. The processing cycle depends on the number of sequence steps.

17.1.2 PLC Functions

17.1.2.1 Built-in PLC Basic Function

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

This CNC has a built-in function for sequence control of machines.
There are the following instructions and functions for creating a sequence program.

- Basic instructions (LD, OR, AND, OUT, etc.)
- Function instructions (data transfer, four arithmetic operations, logical operations, etc.)
- Exclusive instructions (ATC, ROT)
- Applied functions (type conversion functions, arithmetic functions, etc.)

There are ladder language and ST language as languages for writing sequence programs.
As a multi-project function, you can create projects for each machine sequence and manage or start these projects separately with one CNC.

Available project range

M800V	1 to 6 projects (additional specifications for 3 or more projects)
M80V TypeA, M80VW	1 to 3 projects
M80V TypeB	1 to 2 projects

■ Devices

The device range in the following table is the default number of device points set in the project 1 when the multi-project setting or the setting of the number of device points is not performed.

Device	Device No.		Unit	Details
X (*1)	X0 to X1FFF	(8192 points)	1 bit	Input signal to PLC, Machine input, etc.
Y (*1)	Y0 to Y1FFF	(8192 points)	1 bit	Output signal from PLC, Machine output, etc.
M	M0 to M61439	(61440 points)	1 bit	Temporary memory
F	F0 to F2047	(2048 points)	1 bit	Temporary memory (Alarm message interface)
L	L0 to L1023	(1024 points)	1 bit	Latch relay (back up memory)
SM	SM0 to SM2047	(2048 points)	1 bit	Special relay
V	V0 to V511	(512 points)	1 bit	Edge relay
SB	SB0 to SB3FF	(1024 points)	1 bit	Special relay
B	B0 to BDFFF	(57344 points)	1 bit	Link relay
SW	SW0 to SW3FF	(1024 points)	16 bits	Special register
SD	SD0 to SD2047	(2048 points)	16 bits	Special register
T	T0 to T2047	(2048 points)	1 bit/16 bits	Timer (The variable/fixed boundary is set with a parameter.) (*2)
ST	ST0 to ST127	(128 points)	1 bit/16 bits	Integrated timer (The variable/fixed boundary is set with a parameter.) (*5)
C	C0 to C511	(512 points)	1 bit/16 bits	Counter (The variable/fixed boundary is set with a parameter.)
D	D0 to D4095	(4096 points)	16 bits/32 bits	Data register (Register for calculation)
R (*1)	R0 to R32767	(32768 points)	16 bits/32 bits	File register (CNC word interface)
ZR (*1)	ZR0 to ZR13311	(13312 points)	16 bits/32 bits	File register, input/output signals with the PLC, machine input/output, etc.
W	W0 to W2FFF	(12288 points)	16 bits/32 bits	Link register
Z (*3)	Z0 to Z13 / Z0 to Z9	(14 points/10 points)	16 bits	Address index
N	N0 to N7	(8 points)		Master controller nesting level
P (*1)	P0 to P4095	(4096 points)		Label for conditional jump, subroutine call command (*4)
K	K-32768 to K32767			Decimal constant for 16-bit command
	K-2147483648 to K2147483647			Decimal constant for 32-bit command
H	H0 to HFFFF			Hexadecimal constant for 16-bit command
	H0 to HFFFFFFFF			Hexadecimal constant for 32-bit command
E	E±1.17549435-38 to E±3.40282347+38			Real constant for 16-bit command
	E±2.2250738585072014-308 to E±1.7976931348623157+308			Real constant for 32-bit command
$	0 to 32 characters			String constant

(*1) These devices have designated applications. Do not use devices other than those corresponding to the input/output signals with the machine side (input/output signals of the remote I/O unit), even if it is an undefined vacant device.

(*2) Distinction of 10 ms timer and 100 ms timer is performed by command.
(10 ms timer is performed by OUTH command, 100 ms timer is performed by OUT command.)
Timer T can be set in 1 ms increments with a parameter.

(*3) For the Z device, the number of device points depends on the parameter setting.

(*4) The P device has two types of pointers (local and common), and the above points are the total number of points.

(*5) Integrated timer ST can be set in 1 ms or 10 ms increments with a parameter.

■ Parameters that can be used with the built-in PLC

- PLC constants
  This is a data type parameter that can be used with the built-in PLC. There is a basic area and an extended area.
- Bit selection parameters
  This is a bit type parameter that can be used with the built-in PLC. The set data is set in the file register.
- PLC timer
  The timer types include the 10 ms, 100 ms and 100 ms integral types.
  (The timer setting unit can be changed by parameters. (1 ms timer, 1 ms integral timer, and 10 ms integral timer))
  There are two methods for setting the timer setting value: in the PLC program and from the screen. The method can be selected with the parameter.
  Whether to hold the integral timer when the power is turned OFF can also be selected.
- PLC counter
  There are two methods for setting the counter setting value: in the PLC program and from the screen. The method can be selected with the parameter.
  Whether to hold the counter value when the power is turned OFF can also be selected.

■ External key input

By inputting the key data from the built-in PLC, the same operation as when the operator operates the operation board can be done.

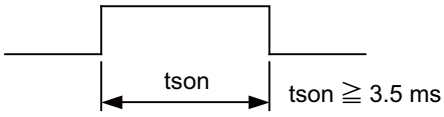
■ Real spindle speed output

The real spindle speed is converted by the signals of the encoder installed on the spindle and is output to the PLC. The output increment is 0.001 r/min.

■ High speed input/output signal

There are signals that can be input and output at a 3.5 ms cycle for high-speed processing.

- Input signal ON time



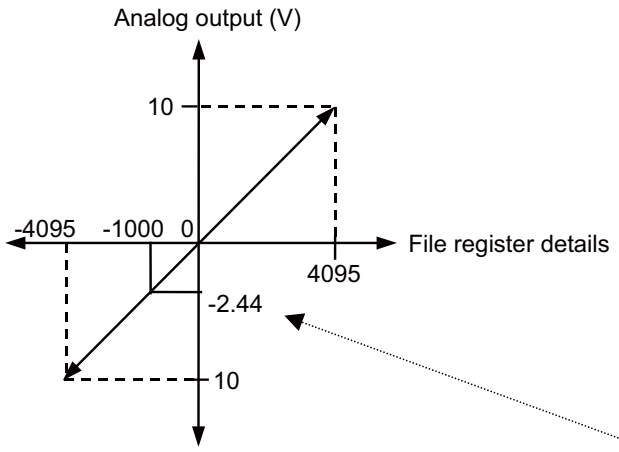
- After the signal output is set in the interface, it can be output to the machine side with delay of up to 3.5 ms. The input also appears on the interface with a 3.5 ms delay.
- The signals used for high-speed processing are assigned with the parameters. Assignment is possible in a continuous 16-point unit.

■ PLC analog voltage control

- Analog output

  When the specified data is put in the file register, the corresponding analog voltage is output from the analog output external connector.

<Relation of file register details and analog output voltage>



Output voltage : -10 V to +10 V (±5%)
Resolution : 2^{-12} (1/4095) x Fullscale (10 V)
Load conditions : 10 kohm resistance load (standard)
Output impedance : 220 ohm

$$\text{Output voltage} = \frac{-1000}{4095} \times 10\ \text{V} = -2.44\ \text{V}$$

Note

(1) For analog output, the hardware corresponding to analog output is required.

Refer to the "PLC Programming Manual" for details.

17.1.2.1.1 Index Modification

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

17.1.2.1.2 Multi-Program [Number of Programs]

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○ 120	○ 120	○ 120	○ 120	○ 60	○ 60	○ 60
L	○ 120	○ 120	○ 120	○ 120	○ 60	○ 60	○ 60

17.1.2.1.3 Multi-Project [Number of Projects Stored]

[M system]

PLC project	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
Number of PLC projects: 2	○	○	○	○	—	—	○
Number of PLC projects: 3	Δ	Δ	Δ	Δ	○	○	—
Number of PLC projects: 6	Δ	Δ	Δ	Δ	—	—	—

[L system]

PLC project	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
Number of PLC projects: 2	○	○	○	○	—	—	○
Number of PLC projects: 3	Δ	Δ	Δ	Δ	○	○	—
Number of PLC projects: 6	Δ	Δ	Δ	Δ	—	—	—

17.1.2.1.4 Function Block (FB)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

17.1.2.1.5 Label Programming

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

17.1.2.1.6 ST Language

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

17.1.2.2 PLC Exclusive Instruction

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

PLC exclusive instruction is provided for certain limited applications, enabling a complex machining process, which is difficult to carry out only by the basic instructions and function instructions.

PLC exclusive instructions include:

■ ATC exclusive instruction (ATC)

With this instruction, ATC, such as magazine index control, tool change by arm, etc. is performed.
ATC exclusive instructions include:

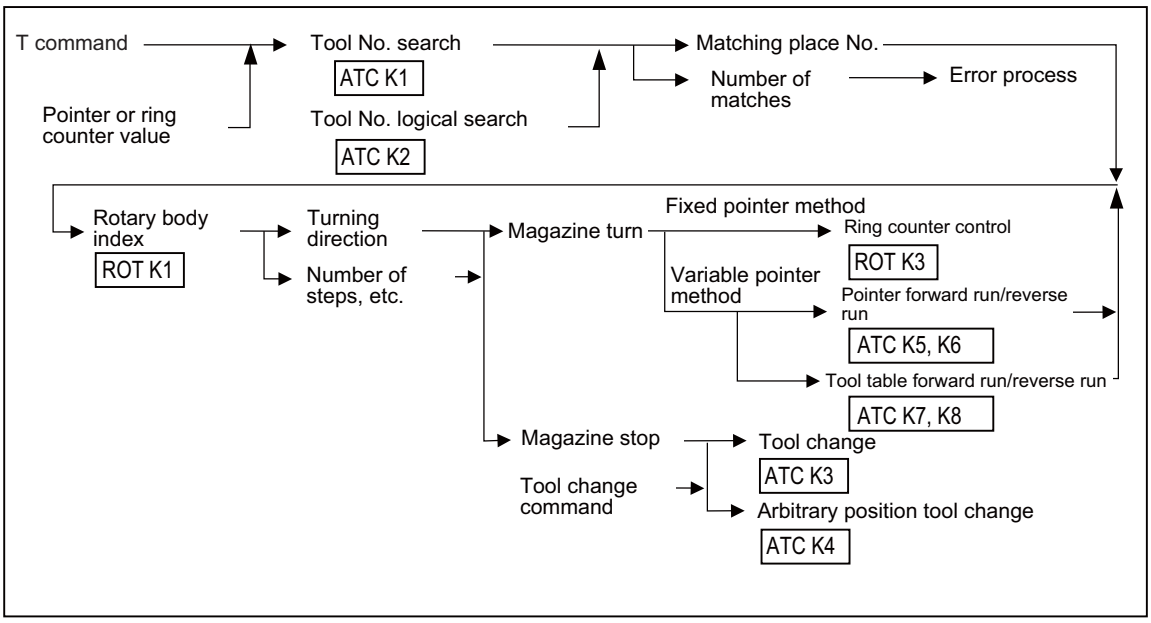
- Tool No. search
- Tool change
- Tool table forward/reverse run
- Pointer (which indicates magazine index position) forward/reverse run
- Tool data read/write

■ Rotary body control instruction (ROT)

This is an instruction to determine the rotary body's target position or rotation direction, or to function as a ring counter. This is used when calculating the rotation direction or number of index steps of the magazine or turret based on the output data figured from ATC exclusive instruction tool No. search processing, or when controlling the rotary body position.

[Using the ATC and ROT instructions]

The order for using the ATC and ROT instructions when T is commanded or tool exchange is commanded is shown below.



17.1.2.3 Enhanced PLC Security Mode

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

This function restricts connections to the built-in PLC from applications such as PLC on-board and GX Developer/GX Works2. This function helps to protect the assets in the built-in PLC.

Two modes are available in this function:

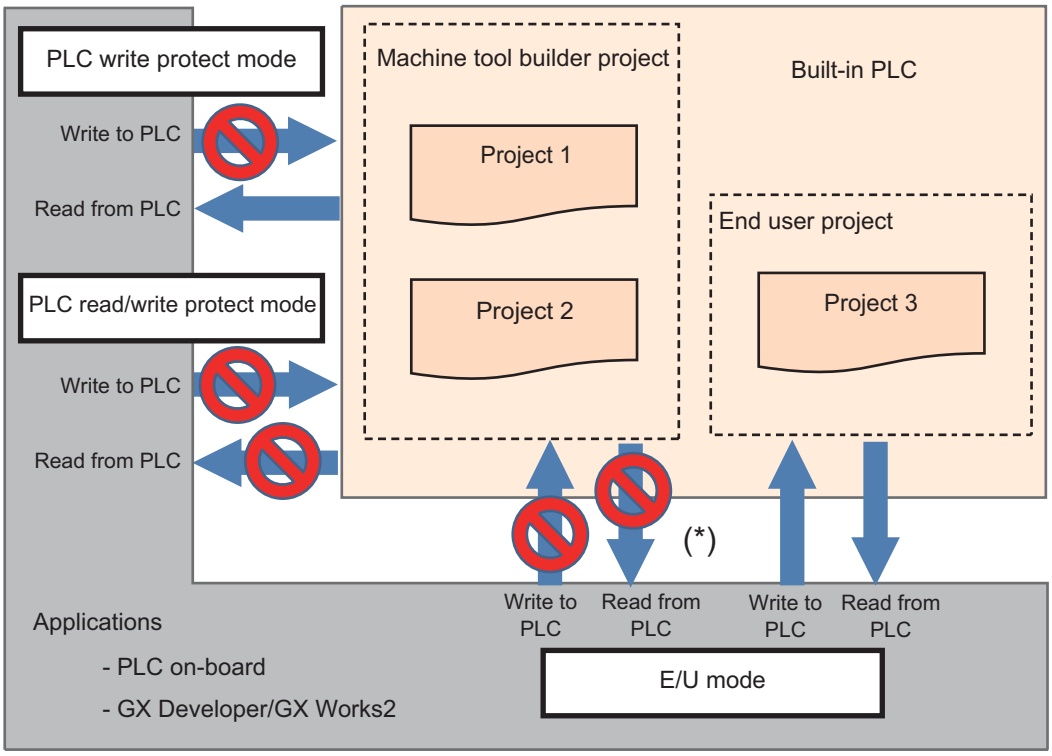
■ PLC write protect mode (inhibits only write operations by applications)

PLC write protect mode prevents users from accidentally overwriting sequence programs and modifying device values when a sequence program is monitored.

■ PLC read/write protect mode (inhibits write and monitoring operations by applications)

PLC read/write protect mode protects sequence programs from being overwritten or read out by an unauthorized user.

While this function is used, one of the multi projects is available as the "end user project". (End user mode (or E/U mode)) Using this project, the end users can create sequence programs to add a new jig and sensor. Restrictions are imposed on the other projects; therefore, there is no risk of modifying or deleting sequence programs created by the machine tool builder.



(*) Read/write operations between the application and the machine tool builder project depend on the enhanced PLC security mode settings.

17.1.3 PLC Support Functions

17.1.3.1 Alarm Message Display

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

The details of the alarms which occur during the sequence (user PLC) process can be displayed on the setting and display unit.
Up to four alarm message displays can be displayed simultaneously on the alarm diagnosis screen. The maximum length of one message is 46 characters.

17.1.3.2 Operator Message Display

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

When a condition to inform the operator of a message occurs, an operator message can be displayed independently of an alarm message.
The maximum length of an operator message on PLC message screen is 60 characters. The maximum number of characters per line is 55, and the message more than 56 characters is displayed on two lines. Up to 4 operator messages can be displayed at a time.

17.1.3.3 Memory Switch (PLC Switch)

The function equivalent to the machine operation switches can be implemented by using the setting display unit of controller. Switches can be turned ON/OFF from PLC switch screen and user PLC. Each switch can be named arbitrarily.

By the PLC switch for reverse, the valid switches by the additional specifications will be reversed.
Even though the PLC switch reverse signals of the disabled switch is turned ON, the input signal is not reversed.

17.1.3.3.1 Memory Switch (PLC Switch) 32 Points

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

32 points of PLC switches can be set on the setting and display unit screen, and the ON/OFF control executed.

17.1.3.3.2 Memory Switch (PLC Switch) 64 Points

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

64 points of PLC switches can be set on the setting and display unit screen, and the ON/OFF control executed.

17.1.3.3.3 Memory Switch (PLC Switch) 96 Points

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	—	—	—
L	Δ	Δ	Δ	Δ	—	—	—

96 points of PLC switches can be set on the setting and display unit screen, and the ON/OFF control executed.

17.1.3.4 Load Meter Display

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

A load meter can be displayed on the setting and display unit.
Up to two axes designated with the built-in PLC such as the spindle load and Z axis load can be displayed as bar graphs on the screen.

17.1.3.5 User PLC Version Display

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

The user PLC version can be displayed in the software list on the Software Configuration screen.

17.1.3.6 Ladder Program Writing During RUN

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

The ladder program can be changed while PLC is running. This function is available, either by GX Developer or PLC onboard edit. (M8 only)

17.1.3.7 PLC Program Protection

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○ (*1)	○ (*1)	○ (*1)	○ (*1)	○ (*1)	○ (*1)	○ (*1)
L	○ (*1)	○ (*1)	○ (*1)	○ (*1)	○ (*1)	○ (*1)	○ (*1)

(*1) Each file

For PLC data protection, the file password can be set to each file of PLC data.
The file password of the PLC data stored in NC can be released temporarily with PLC on-board.

Note

(1) The file password registration/change or cancellation can not be made with PLC on-board. Use GX Developer/GX Works2 for these operation.

The file password has the writing attribute (write protection/permission) and the reading attribute (read protection/permission), and the following operation is prohibited for these attributes.
Write protection: PLC data overwrite and deletion, conversion after editing the ladder
Read/write protection: PLC data overwrite, deletion, read and verification, conversion after editing the ladder

17.1.4 Built-in PLC Capacity

17.1.4.1 Standard PLC Capacity [Number of Steps]

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○ 128K	○ 128K	○ 128K	○ 128K	○ 64K	○ 64K	○ 32K
L	○ 128K	○ 128K	○ 128K	○ 128K	○ 64K	○ 64K	○ 32K

17.1.4.2 Large PLC Capacity: 256 Ksteps

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	—	—	—
L	Δ	Δ	Δ	Δ	—	—	—

17.1.4.3 Large PLC Capacity: 512 Ksteps

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	—	—	—
L	Δ	Δ	Δ	Δ	—	—	—

17.1.5 Machine Contact Input/Output I/F

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

⚠ CAUTION

Follow the remote type machine contact input/output interface described in this manual. (Connect a diode in parallel with the inductive load or connect a protective resistor in serial with the capacitive load, etc.)

Refer to the "Specifications Manual (Hardware)" for details.

17.1.6 Ladder Monitor

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

This function enables the operating status of the sequence circuit to be checked on the controller's setting and display unit.
The ladder monitor functions include the following.

- Circuit monitoring
- Screen stop by monitor stop trigger point
- Entry monitoring
- Decimal-hexadecimal conversion present value monitoring

17.1.7 PLC Development

17.1.7.1 On-Board Development

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

On-board refers generically to the PLC related operations carried out with the CNC unit.
This is the easy-to-use on-board with the enhanced ladder monitoring functions and improved ladder edit usability by enhancing the linkage with the PLC development tools (GX Developer/GX Works2) of Programmable Controller MELSEC series.
User safety sequence of smart safety observation function can also be displayed.
For details on the "smart safety observation function", refer to " 15.5.2 Smart Safety Observation".

For M800V/M80V series, PLC on-board supports labeled and structured projects created with GX Works2. This allows you to view and monitor programs using labels and FBs (function blocks).

17.1.7.2 MELSEC Development Tool (GX Developer)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

This CNC supports the user PLC development environment using the FA engineering software MELSOFT Series (GX Developer), which is the PLC development tool for Programmable Controller MELSEC Series.

17.1.7.3 MELSEC Development Tool (GX Works2)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

This CNC supports the user PLC development environment using the FA engineering software MELSOFT Series (GX Works2), which is the PLC development tool for Programmable Controller MELSEC Series.
GX Works2 is a programming software package (model name:SW1DND-GXW2) for Mitsubishi Electric programmable controller MELSEC series. It enables sequence programs for the CNC to be developed using similar operations to those for MELSEC series. Some of the functions are exclusively for MELSEC series and cannot be used in developing sequence programs for CNC.
GX Works2 Version 1.570U or newer version is recommended for developing sequence programs for the CNC. For details of the functions, refer to each of GX Works2 Operating Manuals.

Note

(1) The software sold separately is necessary.

17.1.8 PLC Parameter

The PLC constants set with the data type and the bit selection parameters set with bit types can be set from the screen as parameters used by the built-in PLC.

17.1.8.1 PLC Constant (150 Points)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

There are PLC constants that can be set with data types as parameters used by the built-in PLC. The set data is set in the R register of the PLC and backed up. If data is set in the R register corresponding to the PLC constant with sequence program MOV commands, etc., the data will be backed up in the PLC constant parameter. However, the display will not change, so enter another screen, and then select this screen again.
The number of PLC constants is 150 points in the basic area, 750 points at the maximum in the extended area, and 300 points in the second extended area.

17.1.8.2 PLC Constant Extension (Up to 750 Points)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

17.1.8.3 Second PLC Constant Extension (300 Points)

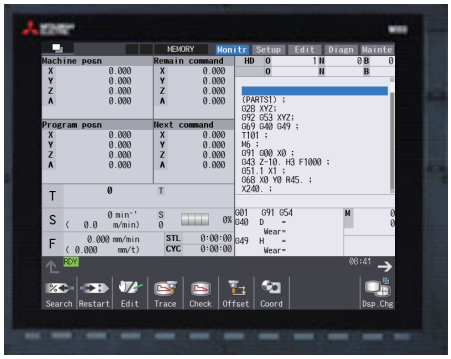
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

17.1.9 GOT Connection

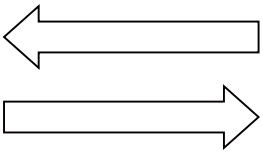
This function connects and communicates between Mitsubishi Electric Graphic Operation Terminal (GOT) and the CNC.
Only GOT2000 series can be connected. Soft GOT 2000 can be connected, too.
There are two connection methods for GOT connection, Ethernet connection and CC-Link connection.
For details, refer to each manual of GOT.

Operation and monitoring from GOT to PLC device of the NC can be performed, and GOT can be used as a sub monitor of peripheral operation panel or NC operation panel.

M8V series



Operation of PLC device



Monitoring PLC device

GOT2000 series



17.1.9.1 Ethernet Connection

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

17.1.9.2 CC-Link Connection

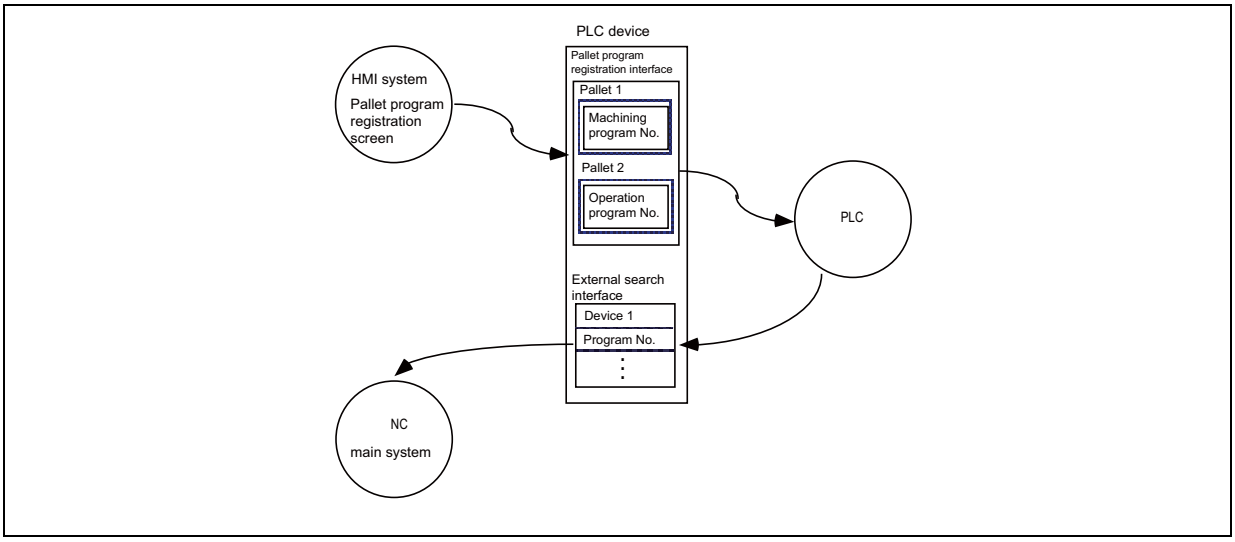
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

17.1.10 Pallet Program Registration

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	—	—	—
L	—	—	—	—	—	—	—

Pallet program function allows machining program to be registered for each pallet of the auto pallet changer (hereinafter referred to as APC) and assists in the machining setups.
Operation search for the registered program will be executed by PLC using the external search function.

Machining program for each pallet is registered at "Pallet program registration" screen in the setup screen group.
Registered program is output to PLC device.
If necessary, execute an external search on PLC ladder by referring to the program No. registered for each pallet.



17.2 Machine Construction

17.2.1 Servo OFF

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

When the "Servo OFF" signal (per axis) is input, the corresponding axis is set in the servo OFF state.
When the moving axis is mechanically clamped, this function is designed to prevent the servomotor from being overloaded by the clamping force.
Even if the motor shaft should move because of some reason in the servo OFF state, the movement amount will be compensated in the next servo ON state by one of the following two methods. (You can select the compensation method using a parameter.)

- The counter is corrected according to the movement amount (follow up function).
- The motor is moved according to the counter and compensated.

When follow up function is designated, the movement amount will be compensated even in the emergency stop state.
The axis is simultaneously set to servo OFF state and the interlock state.

[Mechanical handle]
Even if the servo OFF axis is moved with the mechanical handle with the application of the servo OFF function and follow up function, the position data can be constantly read in and the machine position updated. Thus, even if the axis is moved with the mechanical handle, the coordinate position display will not deviate.

17.2.2 Axis Detachment

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

The axis detachment is a function to shut off the power supply to the motor of an arbitrary control axis (servo axis/spindle) to temporarily disable the axis.

This function is effective for machines as follows.
- Machine which switches between servo motor and spindle motor of spindle/C axis of lathe
- Machine which changes rotary table or attachments of machining center
- When you want to limit the axis configuration that can be commanded depending on the machining process

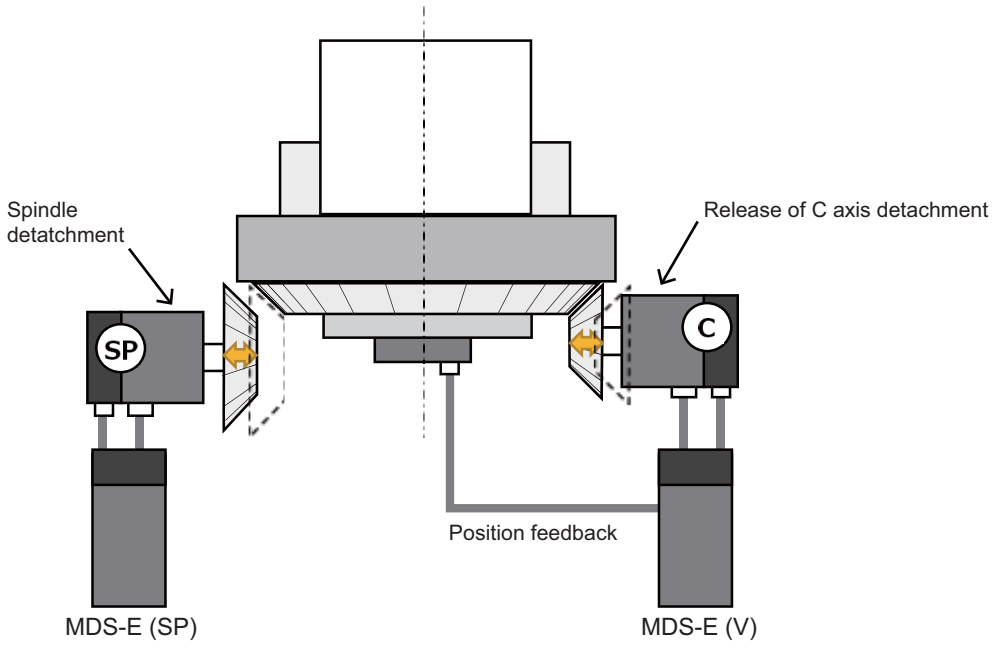
For a control axis during axis detachment, the axis detachment status symbol (">" or "][") is indicated on the right of the coordinate value display counter on the display unit.
Program commands, manual feed commands, spindle forward run (reverse run) commands cannot be performed for an axis during axis detachment. If a movement command is issued to an axis during axis detachment, an alarm (internal interlock) occurs.

The axis detachment function includes control axis detachment 1 and control axis detachment 2 for servo axes, and spindle detachment for spindles.
With the control axis detachment 1, the zero point position is reestablished with the machine configuration where the axis detachment is canceled, then the operation is resumed.
With the control axis detachment 2, the zero point position is kept even after the axis detachment is released. The control can be resumed immediately after the release.

Note

(1) For the absolute position detection system, use the control axis detachment 2 where the zero point is not lost.

[Example of machine configuration for axis detachment (spindle/C axis)]



17.2.3 Synchronous Control

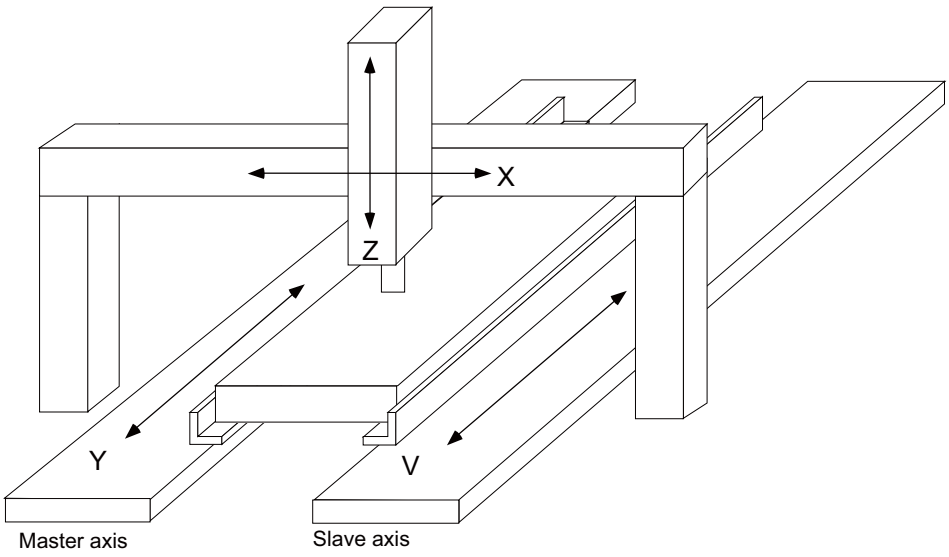
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

The synchronous control is a control method that both master and slave axes are controlled with the same movement command by designated the movement command for the master axis also to the slave axis. This function is assumed to be used in the large machine tool, etc. which drives one axis with two servo motors.
The axis that serves as the reference of synchronization is called "master axis", whereas the axis that follows the master axis is called "slave axis".

Note

(1) The axis detachment function cannot be added to the axes used in the synchronous control.
(2) Refer to the "Programming Manual" for combination with control axis synchronization between part systems.

- The synchronized axes are controlled with travel commands to the master axis.
- One slave axis can be set to one master axis.
- Up to the following sets of the master axes/slave axes can be set for all the part systems in total. 8 sets for M800V Series, 3 sets for M80V Series



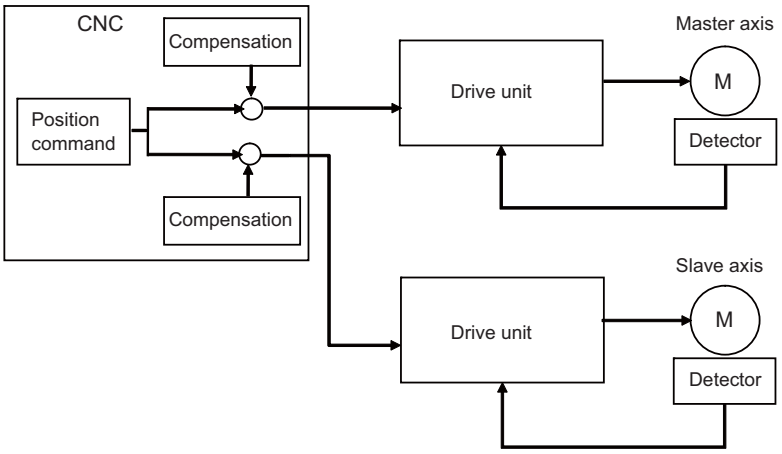
There are 3 types of synchronous control as follows:

Position command synchronous control

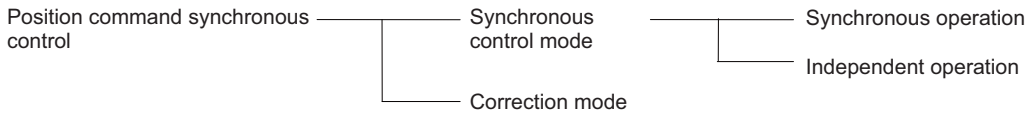
This is used when the machine's rigidity is low.
This conforms the position command of the master axis to that of the slave axis using the synchronous control function of NC and drive those axes in parallel.

This can be used only by changing the NC setting, without changing the settings in the servo drive unit.

[Position command synchronous control diagram]



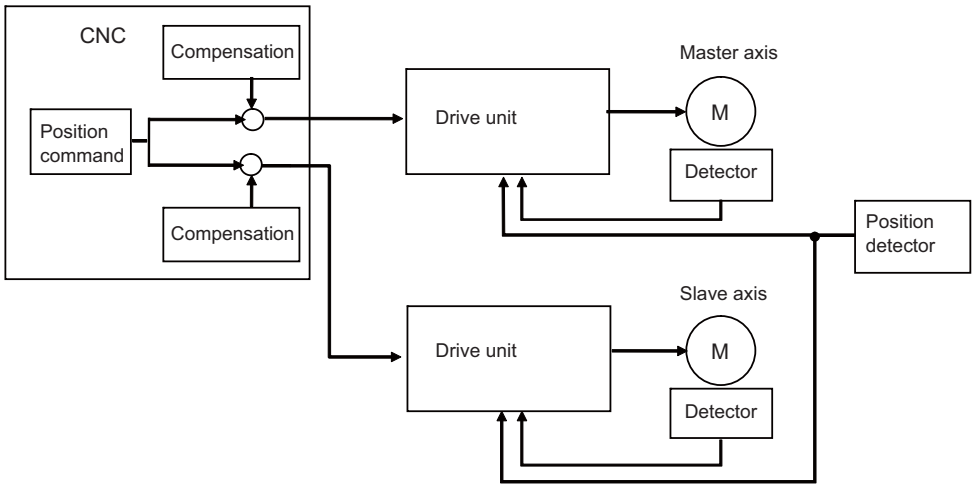
The position command synchronous control has the following configuration:



Speed command synchronous control

This is used when reducing the scale of the full-closed system or when the machine's rigidity, etc. cause the mutual interference to the axes positions.
This drives the master axis and the slave axis in parallel using the common position feedback. Since the feedrate of each axis is controlled by the speed feedback of each axis, the stable control is available.
This can be used when the settings of the NC and the servo drive unit are changed.

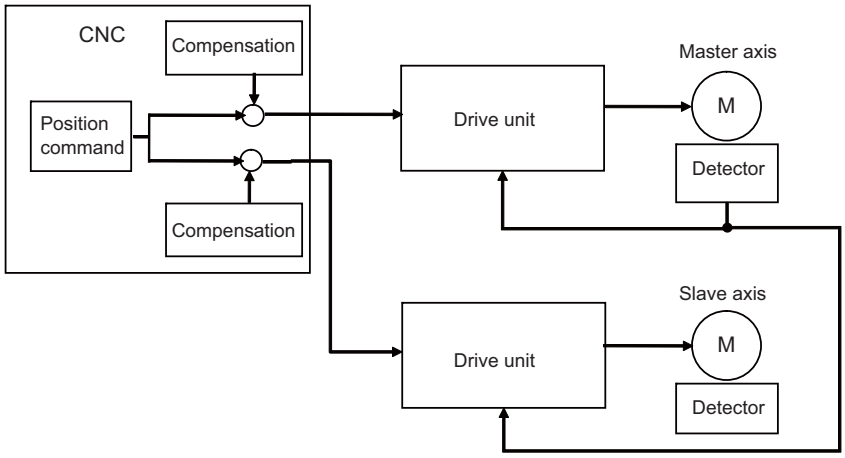
[Speed command synchronous control diagram]



Current command synchronous control

This is used when the machine's rigidity is high.
This drives the master and slave axes in parallel by reference to the speed feedback from the encoder onto the master axis side on the slave axis. Since that speed feedback is not the slave axis's itself, the slave axis easily vibrates. Also, as the slave axis is easily influenced by the disturbance, this is not appropriate for the machine tools which need the positioning accuracy or the interpolation accuracy.
This can be used when the setting of the NC and the servo drive unit are changed.

[Current command synchronous control diagram]



The "compensation" in the Position command synchronous control diagram, Speed command synchronous control diagram, and Current command synchronous control diagram indicates the pitch error compensation, the thermal expansion compensation, the backlash compensation and the external machine coordinate compensation.
The axes which can set the compensation during the synchronous control differ according to the synchronous control type.

There are 2 types of modes as follows:

■ Synchronous control mode

The following two operation methods are available in the synchronous control mode.

- Synchronous operation
  This is a method that both master and slave axes are moved simultaneously with the movement command for the master axis.
- Independent operation
  This is a method that either the master or slave axis is moved with the movement command for the master axis.

■ Correction mode

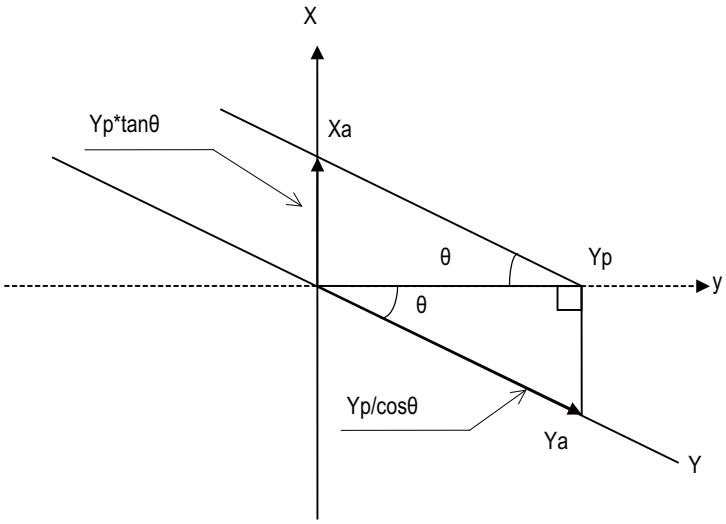
The synchronization is temporary canceled to adjust the balance of the master and slave axes during the synchronous control mode in the machine adjustment. Each axis can be moved separately with the manual handle feed or the arbitrary feed in manual mode. If the operation mode other than the manual handle feed and arbitrary feed in manual mode is applied during the correction mode, the operation error will occur.

17.2.4 Inclined Axis Control

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

Even when the control axes configuring that machine are mounted at an angle other than 90 degrees, this function enables it to be controlled by the same program as that for an orthogonal axis.
The inclination angle is set using a parameter, and axes are controlled using the movement amounts of the axes which are obtained through conversion and compensation using this angle.

[Case in which the X axis serves as the basic axis and the Y axis serves as the inclined axis]



X: Actual X axis
Y: Actual Y axis
y: Programmed Y axis
θ: Inclination angle

Yp, the Y-axis position on the programmed coordinates (on the orthogonal coordinates), is the position of Xa and Ya which are produced by synthesis of X axis and Y axis.
Therefore, the Y-axis (inclined axis) movement amount is expressed by the following formula:
Ya=Yp/cosθ …… (1)
The X-axis (basic axis) movement amount is compensated by the inclined movement of the Y axis, and it is expressed as follows:
Xa=Xp-Yp*tanθ … (2)
The Y-axis (inclined axis) speed is as follows:
"Fa=Fp/cosθ"

Xa, Ya and Fa are the actual movement amounts and speed. Xp, Yp and Fp are the movement amounts and speed on the program coordinates.

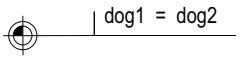
17.2.5 Position Switch

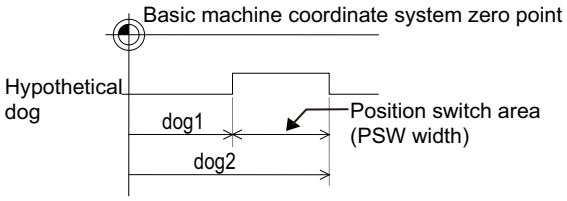
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○ 24	○ 24	○ 24	○ 24	○ 24	○ 24	○ 24
L	○ 24	○ 24	○ 24	○ 24	○ 24	○ 24	○ 24

The position switch (PSW) function provides hypothetical dog switches in place of the dog switches provided on the machine axes so that signals are output to the PLC interface when the machine has reached these hypothetical dog range. The hypothetical dog switches are determined by setting the axis names and coordinate positions indicating hypothetical dog positions (dog1, dog2) in the parameters beforehand. The hypothetical dog switches are known as position switches (PSW). 24 points of the position switches can be set for each part system.
By using the position switch function, the axis is interlocked out of the position switch range to prohibit movement of the target axis. (Position switch interlock function)

Position switch area checking can be performed at high-speed by parameter setting.
In high-speed checking, the parameter determines which is used between the command type machine position or detector feedback position for area checking by each position switch.

[Example of dog1, dog2 settings and execution (for linear axes)]

dog1, dog2 settings	dog1, dog2 positions	Details
dog1 < dog2		Signal is output between dog1 and dog2
dog1 > dog2		Signal is output between dog2 and dog1
dog1 = dog2		Signal is output at the dog1 (dog2) position



17.2.7 Index Table Indexing

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

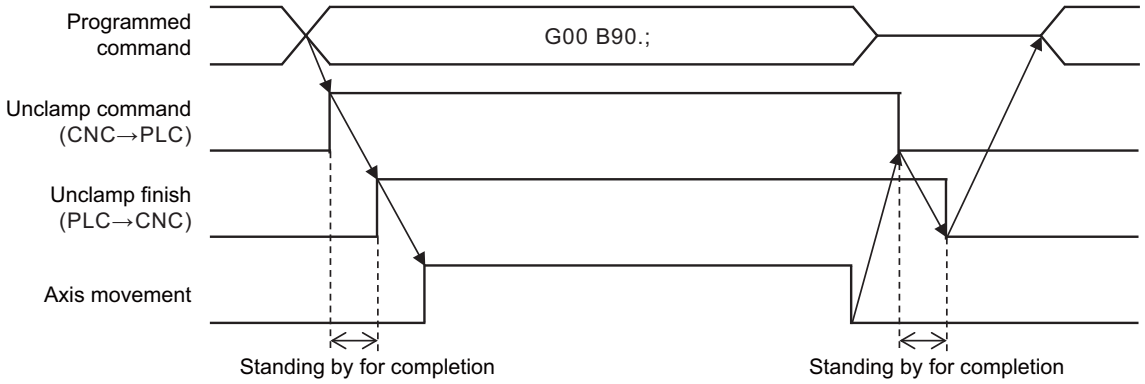
The indexing of the index table can be performed by setting the index axes.
Programming is facilitated because, in terms of the index commands, only the indexing angle need to be designated using the address of the programmed axis serving as the index setting axis, and there is no need to designate special M codes for clamping and unclamping the table.

Clamp operation is performed by setting the "Unclamp command" signal to OFF (type A) or is performed with the clamp signal (type B).

Type A

(1) Designate the movement commands (absolute or incremental) for the selected axis using a program.
(2) The "Unclamp command" signal is now output prior to the axis movement.
(3) When the axes are unclamped, set the "Unclamp completion" signal to ON with the PLC. (Set the signal to ON after performing required process such as servo ON or the process to release the clamp.)
(4) After checking the "Unclamp completion" signal, the designated axis starts moving.
(5) Upon completion of the movement, the "Unclamp command" signal is set to OFF.
(6) Clamp the axes and set the "Unclamp completion" signal to OFF with the PLC. (Set the signal to OFF after performing required process such as in-position check, servo OFF or the clamp process.)
(7) After checking that the "Unclamp completion" signal is OFF, processing of the next block is initiated.

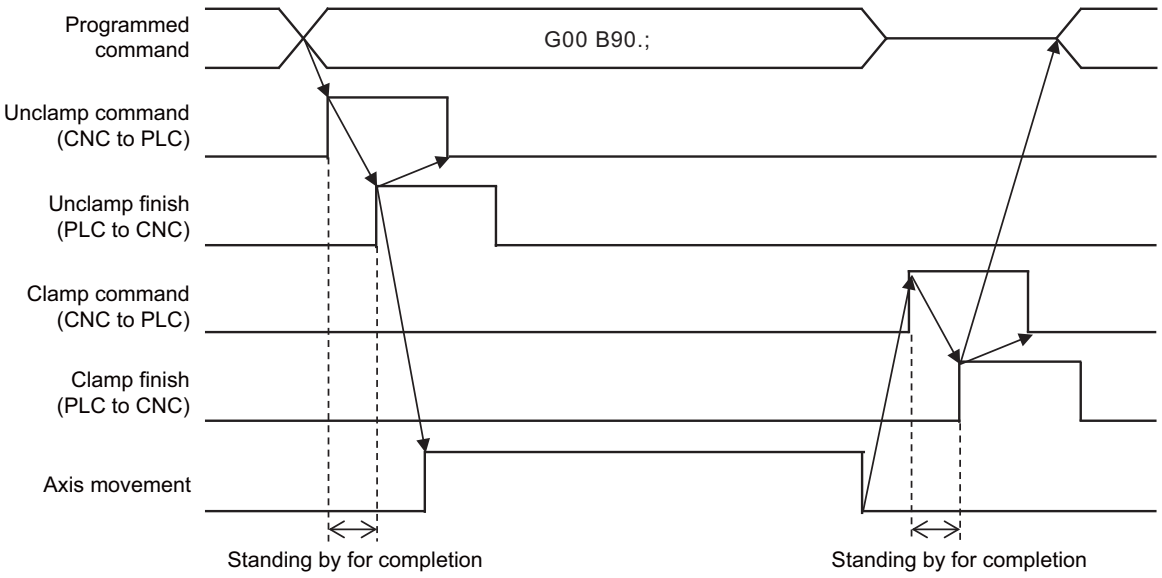
[Operation timing chart for type A]



Type B

(1) Designate the movement commands (absolute or incremental) for the selected axis using a program.
(2) The "Unclamp command" signal is now output prior to the axis movement.
(3) When the axes are unclamped, set the "Unclamp completion" signal to ON with the PLC.
(Set the signal to ON after performing required process such as servo ON or the process to release the clamp.)
(4) After checking the "Unclamp completion" signal, set the "Unclamp command" signal to OFF and the designated axis starts moving.
(5) Set the "Unclamp completion" signal to OFF with the PLC.
(6) Upon completion of the movement, the "Clamp command" signal is set to ON.
(7) Clamp the axes and set the "Clamp completion" signal to ON with the PLC.
(Set the signal to ON after performing required process such as in-position check, servo OFF or the clamp process.)
(8) After checking that the "Clamp completion" signal is ON, set the "Camp command" signal to OFF and processing of the next block is initiated.
(9) Set the "Clamp completion" signal to OFF with the PLC.

[Operation timing chart for type B]



17.2.8 Tool Length Compensation Along the Tool Axis

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	—	—	—
L	Δ(*1)	Δ(*1)	Δ(*1)	Δ(*1)	—	—	—

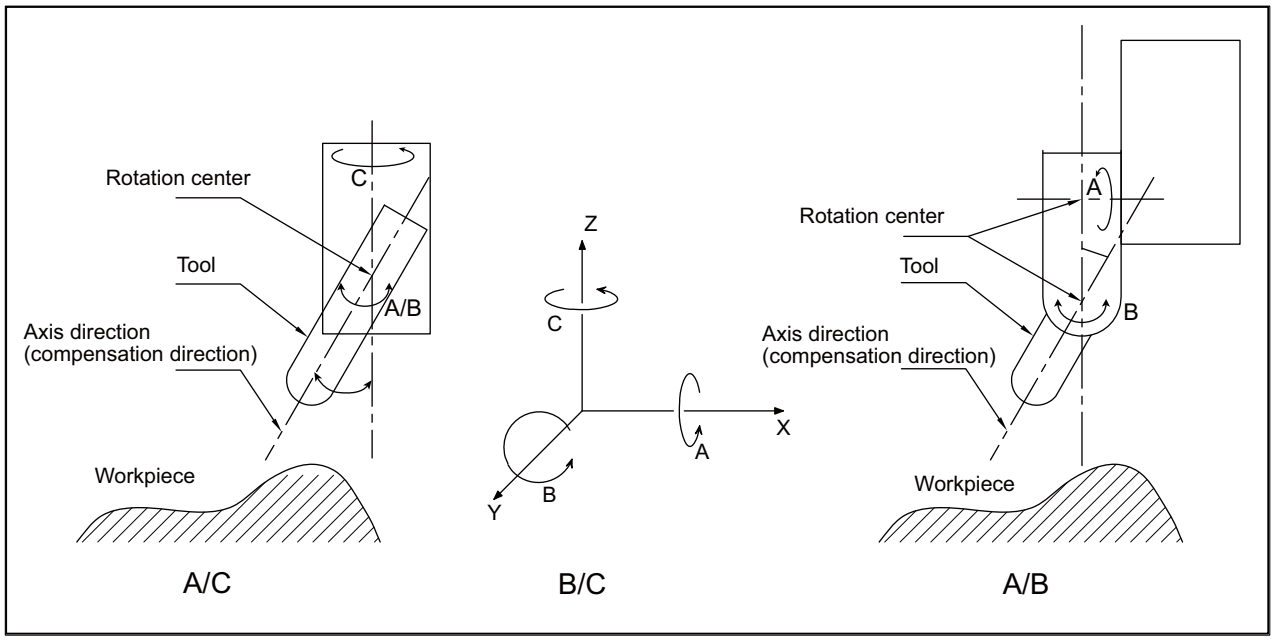
(*1) This function is available during program format switch.

■ Changing the tool length compensation in tool axis direction and compensation amount

Even if the tool axis direction is not the Z axis direction because the rotary axis is rotated, the tool can be compensated in the tool axis direction. By setting the deviation of the tool length value set when the program was created and the actual tool length as a compensation amount, the program can be made more flexible. This is especially effective for programs with many rotary axis movement commands.
The tool length compensation amount in the tool axis direction can be changed by rotating the manual pulse generator when the tool length compensation along tool axis mode and tool length compensation amount along the tool axis change mode are valid.

■ Machine configuration

The tool length compensation in tool axis direction is carried out in respect to the direction of the tool nose axis (rotary axis). The axis which determines the compensation direction is designated with the parameters as a combination of the Z-axis rotation C axis (spindle) and X-axis rotation A axis or Y-axis rotation B axis.

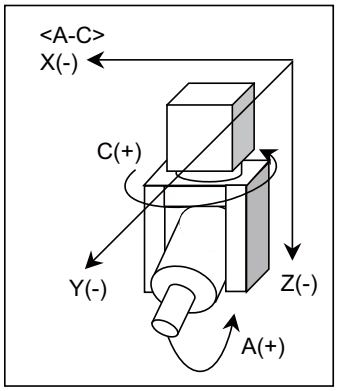


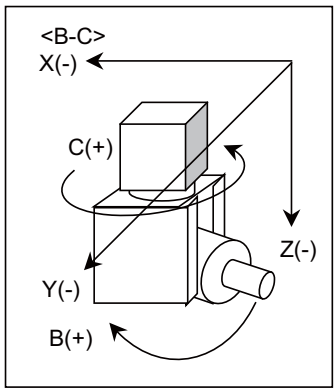
17.2.9 Tool Handle Feed & Interruption

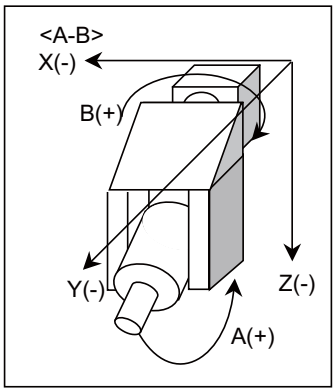
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	—	—	—
L	—	—	—	—	—	—	—

The tool handle feed & interrupt function makes it possible to move the axis with the manual pulse generator in the tool axis direction, tool diameter direction X and tool diameter direction Y within the hypothetical coordinate system over the tool axis.

Normal handle feed and interrupt operation is applied when this function is not provided.







This is also effective for a machine configuration having the tool tilt 1 axis and table tilt 1 axis.

17.2.10 Tool Center Coordinate Display

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	—	—	—
L	—	—	—	—	—	—	—

The tool center coordinates, handle interrupt amount (tool axis movement), table/workpiece installation, machine position coordinates in the inclined surface coordinate system, mechanical axis angle and tool center point speed are displayed during the tool center point control function, tool length compensation along tool axis function, and tool handle feed / interrupt function (tool axis direction handle feed, tool handle interrupt, tool diameter direction handle feed, nose center rotation handle feed).
The path using the tool center coordinate position is drawn on the Graphic Trace screen.
One of the following functions is required to validate this function.

- Tool length compensation along the tool axis
- Tool handle feed & interruption
- Tool center point control
- 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

17.2.11 Tool Center Point Control

17.2.11.1 Tool Center Point Control (G43.4/G43.5)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ(*1)	Δ	Δ(*1)	○ (*1)	○ (*1)	—
L	—	—	—	—	—	—	—

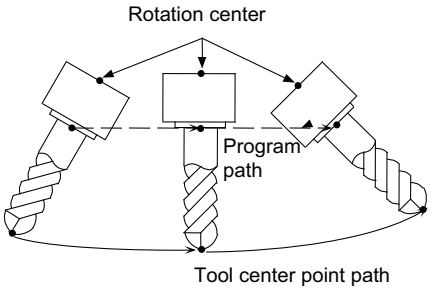
(*1) Restrained to 4-axis simultaneous contouring

The tool center point control function controls so that the position command in the machining program is at the tool center point on the coordinate system (table coordinate system) which rotates together with the workpiece. This function can be applied to the following types of machines.

Tool tilt type	A machine with two rotary axes set on the head.
Table tilt type	A machine with two rotary axes set on the table.
Combined type	A machine with one rotary axis set on the tool and another on the table.

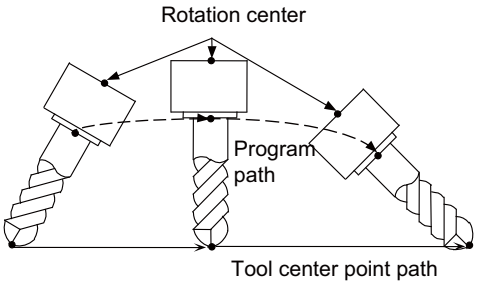
■ Tool tilt type

Tool center point control is OFF
Tool length compensation along tool axis is ON



Controls so that the holder center point moves straight.

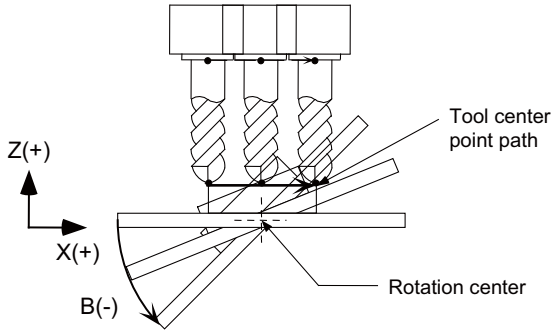
Tool center point control is ON



Controls so that the tool center point draws a straight line.

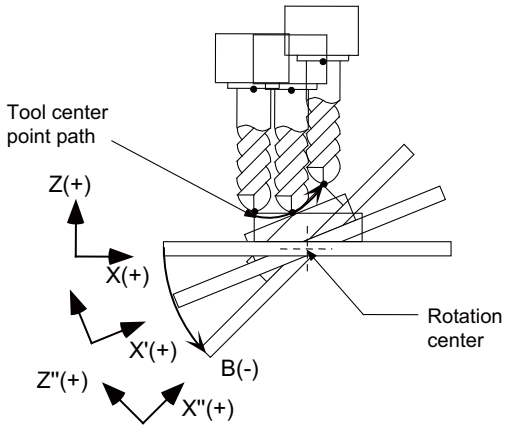
■ Table tilt type

Tool center point control is OFF
Tool length compensation along tool axis is ON



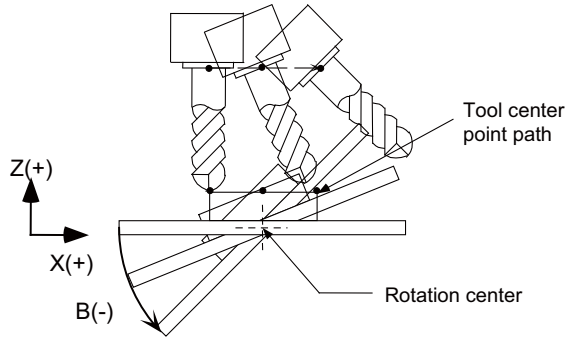
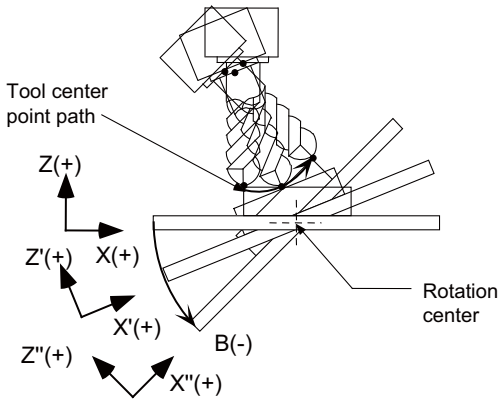
Controls so that the holder center point is at the workpiece coordinate system's position.

Tool center point control is ON



Controls so that the tool center point is at the table coordinate system's position.

■ Combined type

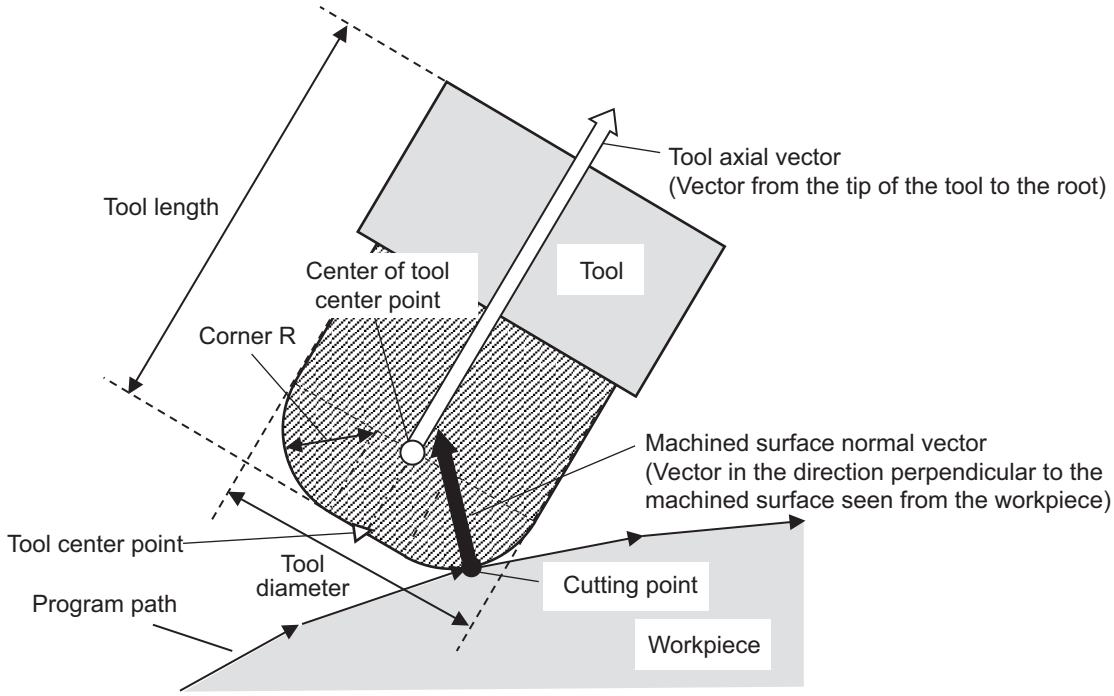
Tool center point control is OFF Tool length compensation along tool axis is ON	Tool center point control is ON
	
Controls so that the holder center point is at the workpiece coordinate system's position.	Controls so that the tool center point is at the table coordinate system's position.

For the machine configuration required to use this function, refer to "18 Appendix 1: Functions with Advanced Machining".

17.2.11.2 Tool Cutting Point Control Command (G43.8/G43.9)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	—	Δ	—	—	—	—
L	—	—	—	—	—	—	—

Tool cutting point control is a function to control the point (cutting point) where the tool comes into contact with the workpiece so that it moves at the path and speed specified by the machining program.
NC automatically compensates the tool length and tool shape (tool radius, corner R). Therefore, machining can be performed without changing the machining program to change tools. This has the advantage of improving the diversion of machining programs.

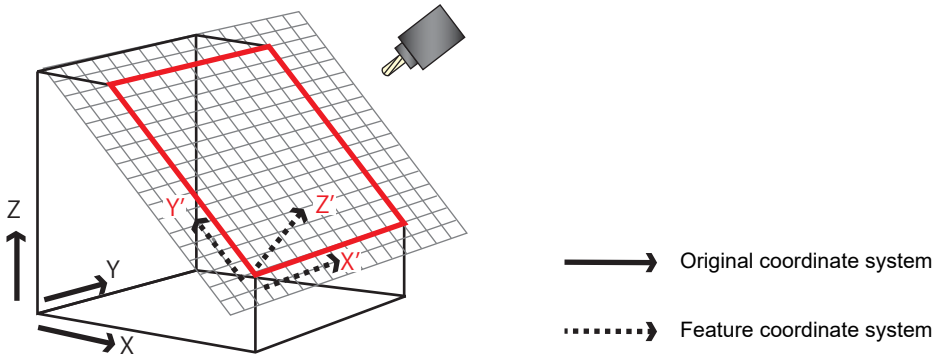


To use this function, additional specifications of both "tool cutting point control" and "tool center point control" are required. For the machine configuration required to use this function, refer to "18 Appendix 1: Functions with Advanced Machining".

17.2.12 Inclined Surface Machining Command

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	—
L	Δ	Δ	Δ	Δ	○	○	—

This function is to define the new coordinate system (called the "feature coordinate system") which was created by rotation and reference position translation of the X, Y, Z axes of the original one (before the inclined surface machining command was applied). With this function, you can define an arbitrary plane in a space and issue normal program commands to this plane in machining.
Also, the tool axis can automatically be controlled to the + Z direction of the newly defined feature coordinate system. Since the feature coordinate system will be re-created in accordance with the tool axis direction, machining programs can be developed without the need to consider the direction of the feature coordinate system and rotation of the tool axis.



The feature coordinate system is defined in the following method.

- Designation with Euler angle.
- Designation with the roll angle, pitch and yaw angle.
- Designation with the three points on the plane.
- Designation with two vectors.
- Designation with a projection angle.
- Designation with the registered machining surface selection
- Designation with the tool axis direction.

For the machine configuration required to use this function, refer to "18 Appendix 1: Functions with Advanced Machining".

17.2.13 Simple Inclined Surface Machining Command

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	—	—	—	—	—
L	Δ	Δ	Δ	Δ	○	○	○

In a lathe with three orthogonal axes and one rotary axis on the tool side, this function enables simultaneous 4-axis control milling on an inclined surface along workpiece end face.
To facilitate machining on an inclined surface, this function uses three commands: simple inclined surface control (G176), simple tool center point control (G174), and tool axis direction control (G53.1) as follows. The tool axis direction can be turned at right angles to the inclined surface by issuing the tool axis direction control command (G53.1) after the simple inclined surface control command (G176). When G174 is given after G176, the direction of tool length is compensated in accordance with the tool angle.
For a machine configured with a turret, this function allows you to use any tool on the turret to carry out simple inclined surface machining.

■ Simple inclined surface control (G176)

G176 enables a new coordinate system (called "feature coordinate system") to be defined. Feature coordinate system is obtained by rotating and parallel translating a plane using the X, Y and Z axes configuring the workpiece coordinate system as a rotary axis. Consequently, there is no need to calculate the coordinate position in accordance with the inclined surface angle. This simplifies creation of a machining program.

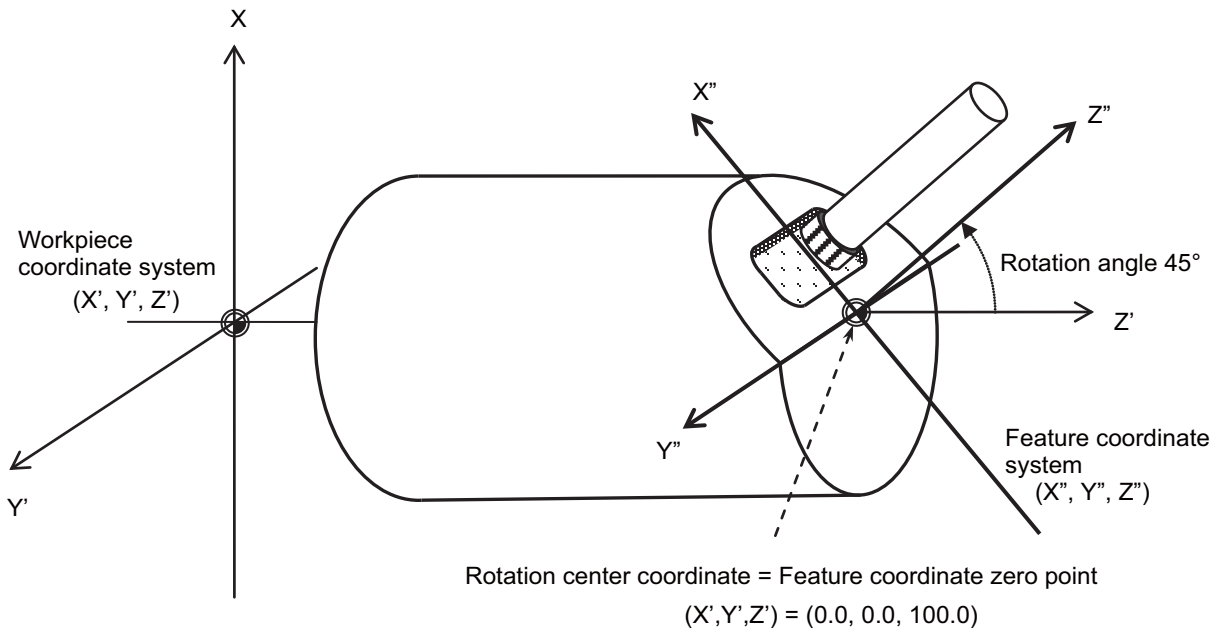
■ Simple tool center point control (G174)

G174 allows a tool length offset in the tool axis direction even when the tool axis direction is not parallel to the orthogonal coordinate system after rotation of the rotary axis. Consequently, the nose of the tool mounted on the rotary axis can be kept in the programmed position at all times. This simplifies creation of a complicated-shape machining program.

■ Tool axis direction control (G53.1)

G53.1 aligns the tool with the + Z direction of the feature coordinate system. This eliminates the need for you to mind the feature coordinate system's direction or tool axis' rotation direction, resulting in greater ease of programming.

[Feature coordinate system]



■ Command format

Simple inclined surface control start

G176 X__Z__D__ ; When rotating K-I plane about the axis parallel to the J axis	
G176 X__Y__D__ ; When rotating I-J plane about the axis parallel to the K axis	
G176 Y__Z__D__ ; When rotating J-K plane about the axis parallel to the I axis	
X, Y, Z	: Feature coordinate system zero point (Rotation center position) (-99999.999 to 99999.999 (mm)) (-9999.9999 to 9999.9999 (inch))
D	: Rotation angle (-359.999 to 359.999 (°))

Simple inclined surface control cancel

G69.1;

Tool axis direction control

G53.1 R__ ;	
R	: Tool axis rotation angle compensation amount (-359.999 to 359.999 (°))

Simple tool center point control start

G174 R__ ;	
R	: Tool axis rotation angle compensation amount (-359.999 to 359.999 (°))

Simple tool center point control cancel command

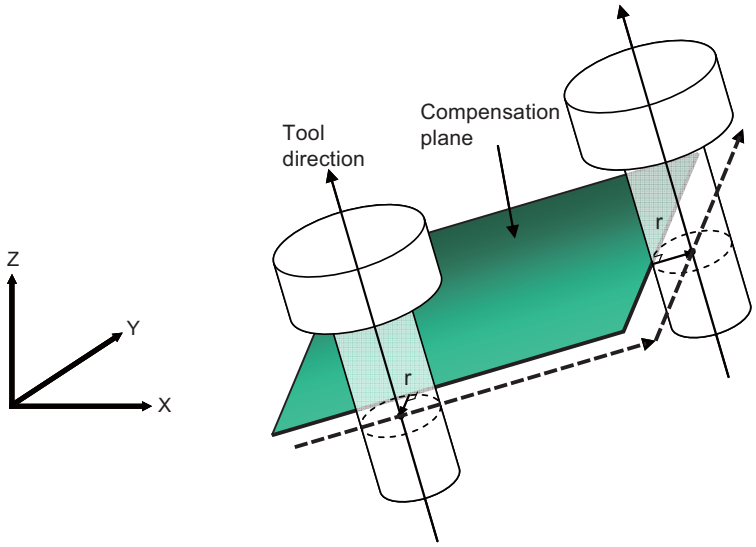
G175;

17.2.14 3-Dimensional Tool Radius Compensation (Tool's Vertical-Direction Compensation)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	—	—	—
L	Δ(*1)	Δ(*1)	Δ(*1)	Δ(*1)	—	—	—

(*1) This function is available during program format switch.

This function is to compensate the tool radius of the machine with two rotary axes, in accordance with the change of the workpiece direction and inclination of the tool due to the movement of the rotary axis. The 3-dimensional tool radius compensation was realized by searching the tool path on the workpiece by the program command and calculating the compensation vector on the plane (compensation plane) which is vertical to the path in the tool direction.



r : Compensation amount

: Tool center path

: Program path

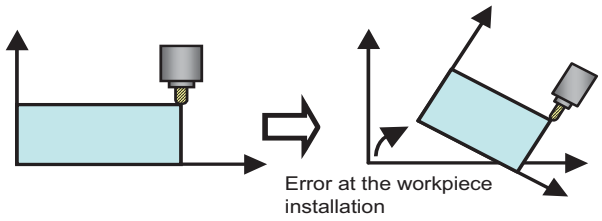
When the 3-dimensional tool radius compensation (tool's vertical-direction compensation) is commanded while the specification is not added, an alarm will occur.
For the machine configuration required to use this function, refer to "18 Appendix 1: Functions with Advanced Machining".

17.2.15 Workpiece Installation Error Compensation

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	—	Δ	—	—	—	—
L	Δ(*1)	—	Δ(*1)	—	—	—	—

(*1) This function is available during program format switch.

This function is for the 5-axis machine. This compensates the error when a workpiece is placed off the workpiece coordinate system to enable machining according to the program. In this function, a new coordinate system with the workpiece as its reference position will be defined (called "workpiece installation coordinate system") and the program will be executed in this new coordinate system.



For the machine configuration required to use this function, refer to "18 Appendix 1: Functions with Advanced Machining".

17.2.16 3-Dimensional Manual Feed

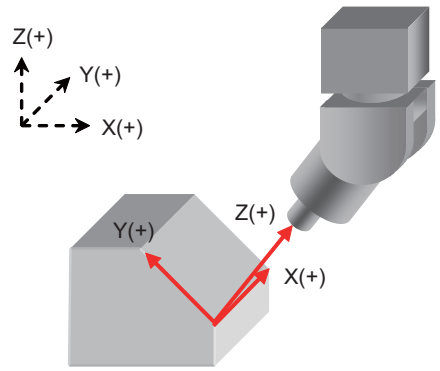
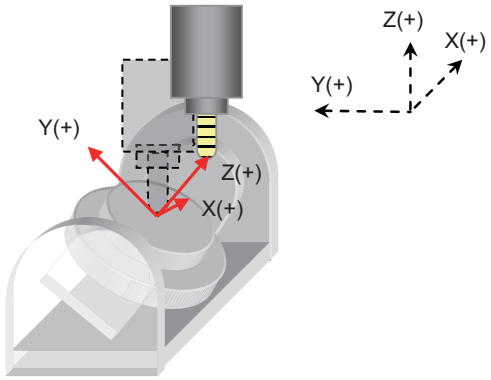
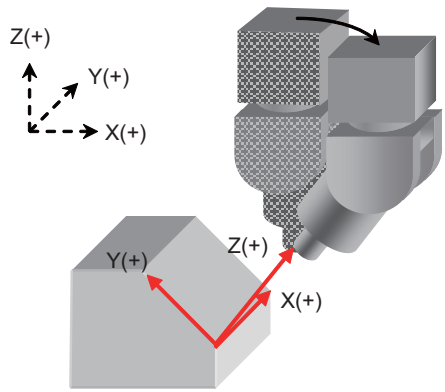
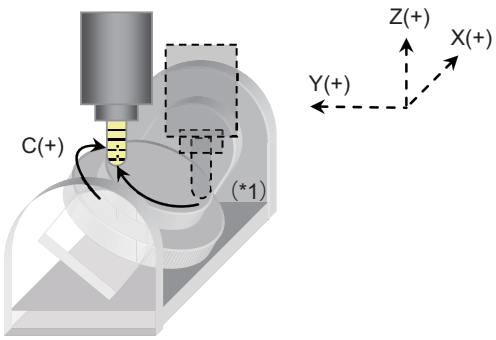
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	—
L	Δ	Δ	Δ	Δ	○	○	—

By selecting the hypothetical coordinate system to be machined, axis can be moved with manual feed (JOG, HANDLE or INCREMENTAL) in the coordinate system with this function.
If a movement command on the hypothetical coordinate is issued, it can be easy to setup because multiple axes is moved by NC according to the tool angle or the inclination of the table.

This function consists of the hypothetical coordinate feed and tool center point rotation.
Movement command for the linear axis operates as the hypothetical coordinate feed and manual feed will be carried out on the hypothetical coordinate.
Movement command for the rotary axis operates as the tool center point rotation and manual feed will be carried out with the rotary axis and 3 linear axes which are issued a movement command to keep position relations of the workpiece and tool tip.
This function can be used on the machine whose rotary axis is installed with ISO polarity and the machine whose axis is installed with non-ISO polarity. However, the orthogonal axis needs to be installed with ISO polarity.(*1)
Note that this function can be also used on the machine with a tilted rotary axis.

(*1) In the machine coordinate system (right-hand orthogonal coordinate system) defined in ISO standard, the rotation turning to the right-handed screw is handled as ISO polarity and the one turning to the left-handed screw is handled as non-ISO polarity.

Standard manual feed is applied when this function is not provided.

	<Tool tilt B-C axis>	<Table tilt A-C axis>
Linear axis: Hypothetical coordinate feed		
Rotary axis: Tool center point rotation		 (*1) Keep the position of the workpiece and tool tip along C axis rotation.

Original coordinate system

Hypothetical coordinate system

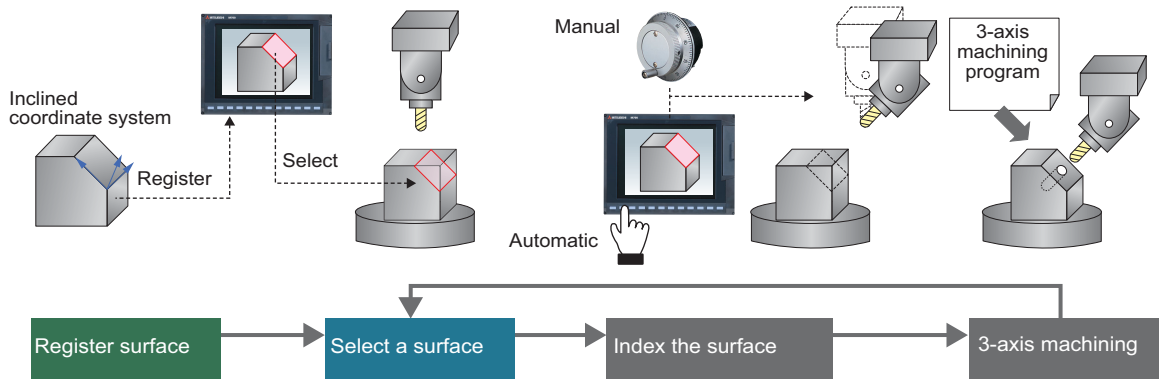
17.2.17 R-Navi

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	—
L	—	—	—	—	—	—	—

This function allows you to perform index machining (including inclined surface and multiple-surface machining) simply and smoothly using a rotary axis.
To enable index machining, register a feature coordinate system (coordinate origin and coordinate axial direction) along the machining surface in advance, and select the registered surface from the operation screen. Then the tool is able to move along the coordinate system to perform index machining.
As shown below, the machining surface selected from the screen is not affected by whether the surface indexing is complete or not.
The function is effective for the machine configuration with the right-hand orthogonal coordinate system defined in ISO standard.
The R-Navi function enables you to use "Inclined machining surface" and "3-dimensional manual feed".

Features of this function are as follows;

- Various and simple machining surface registration (Feature coordinate system registration)
  - Possible to register multiple surfaces (Up to 16 surfaces per workpiece, and up to 10 workpieces)
  - Guide drawing which helps to set the coordinate system along the machining surface
  - Feature coordinate system setting without dependence on the workpiece installation position.
  - 3D workpiece graphics which enables a visual check of the registered surface.
- Easy-to-understand machining surface selection
  - An operator is able to select a surface while observing the 3D workpiece graphics.
- Automatic indexing and manual indexing of the selected machining surface
  - R-Navi automatically executes surface indexing to set the tool to be perpendicular to the selected surface.
  - Manual indexing is also available.
  - The coordinate system is linked with rotation of the rotary axis (no need to calculate the coordinate origin after the rotation).
- A program for 3-axis machining can be used without modification (G68.2 command-less inclined surface machining).
  - Able to work on the coordinate system defined along the selected surface (by automatic or manual operation)
  - The coordinate system is held even when NC reset is input (no need to re-define the coordinate system).



■ Command format

The R-Navi setup parameters can be configured from a machining program.

[Workpiece registration and setting]

G69 ;	**Cancel the selected machining surface**
G10 L110 ;	**Start setting workpiece data**
Q__ <__> F__ C__ R__ X__ Y__ Z__ I__ J__ K__ ;	**Data setting**
G11 ;	**End data setting**

Q	Workpiece registration No. (1 to 10)
< >	Workpiece name (maximum 20 characters)
F	Workpiece shape
C	Basic coordinate system
R	Basic No. (0 to 8)
X/Y/Z	Workpiece size (0.000 to 99999.999)
I, J, K	Workpiece shift (-99999.999 to 99999.999)

[Machining surface registration and setting]

G69 ;	**Cancel the selected machining surface**
G10 L111 ;	**Start setting machining surface data**
P0 Q__ D__ <__> X__ Y__ Z__ A__ ;	**Machining surface setting**
P1 M__ B__ C__ E__ F__ H__ I__ ;	**Designate coordinate axial direction for 1st axis**
P2 M__ B__ C__ E__ F__ H__ I__ ;	**Designate coordinate axial direction for 2nd axis**
G11 ;	**End data setting**
G68.2 P10 Q__ D__;	**Select the registered machining surface**

Command address of machining surface registration

P	Machining surface registration
Q	Workpiece registration No. (1 to 10)
D	Machining surface registration No. (2 to 17)
< >	Machining surface name (maximum 15 characters)
X/Y/Z	Feature coordinate origin (-99999.999 to 99999.999)
A	Coordinate axis selection

Command address of coordinate axial direction designation

P	Coordinate axial direction axis designation
M	Coordinate axial direction designation method
B, C, E, F, H, I	Coordinate axial direction setting (-99999.999 to 99999.999)

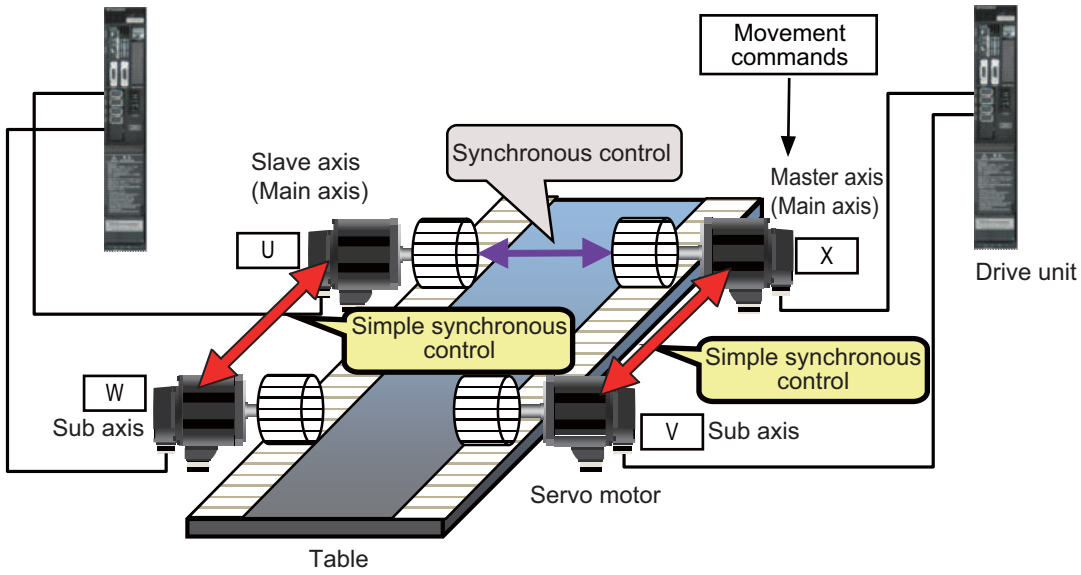
17.2.18 Simple Synchronous Control

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	—
L	—	—	—	—	—	—	—

This function is an extension of the synchronous control function. This function enables an axis to be synchronized with the axis previously defined as master axis for the synchronous control, and another axis to be synchronized with the slave axis. Backlash can be restrained by distributing load torque imposed on one axis to two servo motors. When the synchronous control and the simple synchronous control are combined, one axis can be driven with four servo motors. An axis to be synchronized with is referred to as a "main axis" and an axis which follows the main axis is referred to as a "sub axis".
This is a control method in which the main axis and the sub axis are controlled in the same position by executing a position command for the main axis to the sub axis as well.
The main points of the method are as follows:

- The sub axes can be set only for the master axis and the slave axis for the synchronous control.
- The sub axis is controlled with the movement command for the main axis.
- Only one sub axis can be set for the main axis.
- The main axis and the sub axis are controlled at the same position. Therefore, the main axis and the sub axis can only be used together, while either of the axes can be used singly for single operation and correction mode for the synchronous control.
- For simple synchronous control, synchronization error between the main axis and the sub axis is not monitored.
- For the main axis and the sub axis, specify the speed command synchronous control or the current command synchronous control.

[Example of use of the simple synchronous control]

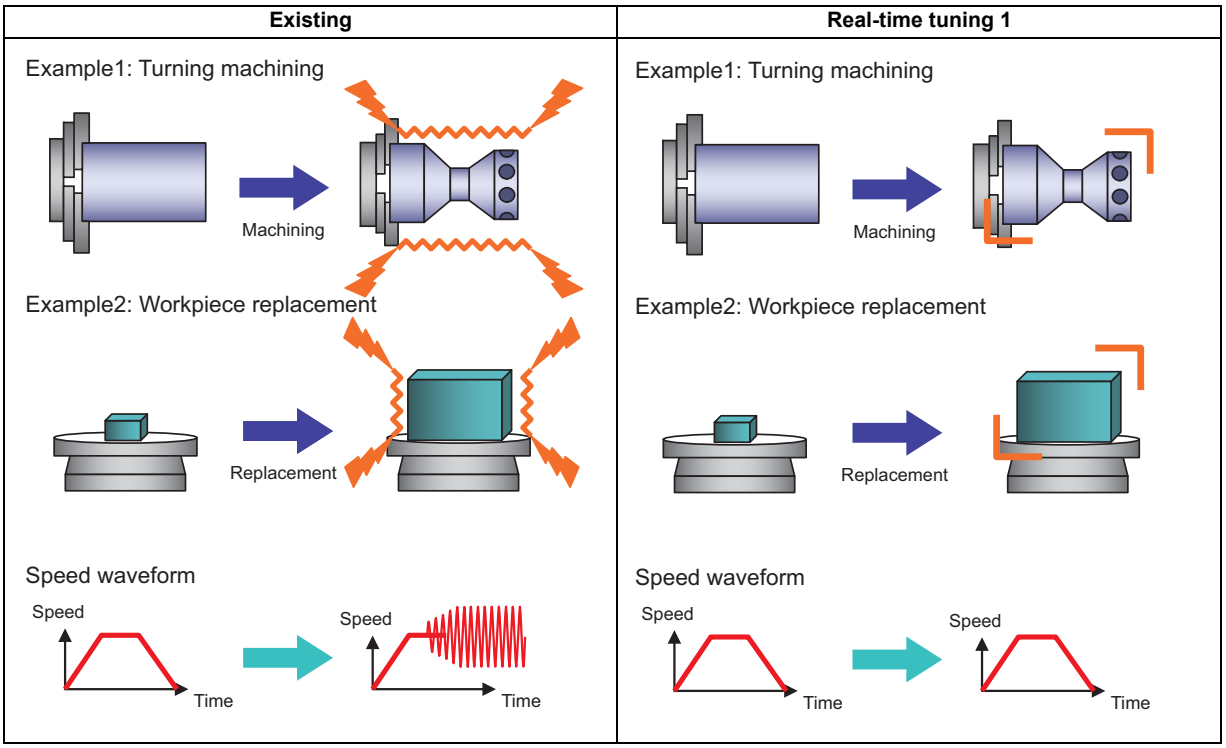


17.2.20 Real-Time Tuning

17.2.20.1 Real-Time Tuning 1 (Speed Gain)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	—
L	Δ	Δ	Δ	Δ	○	○	—

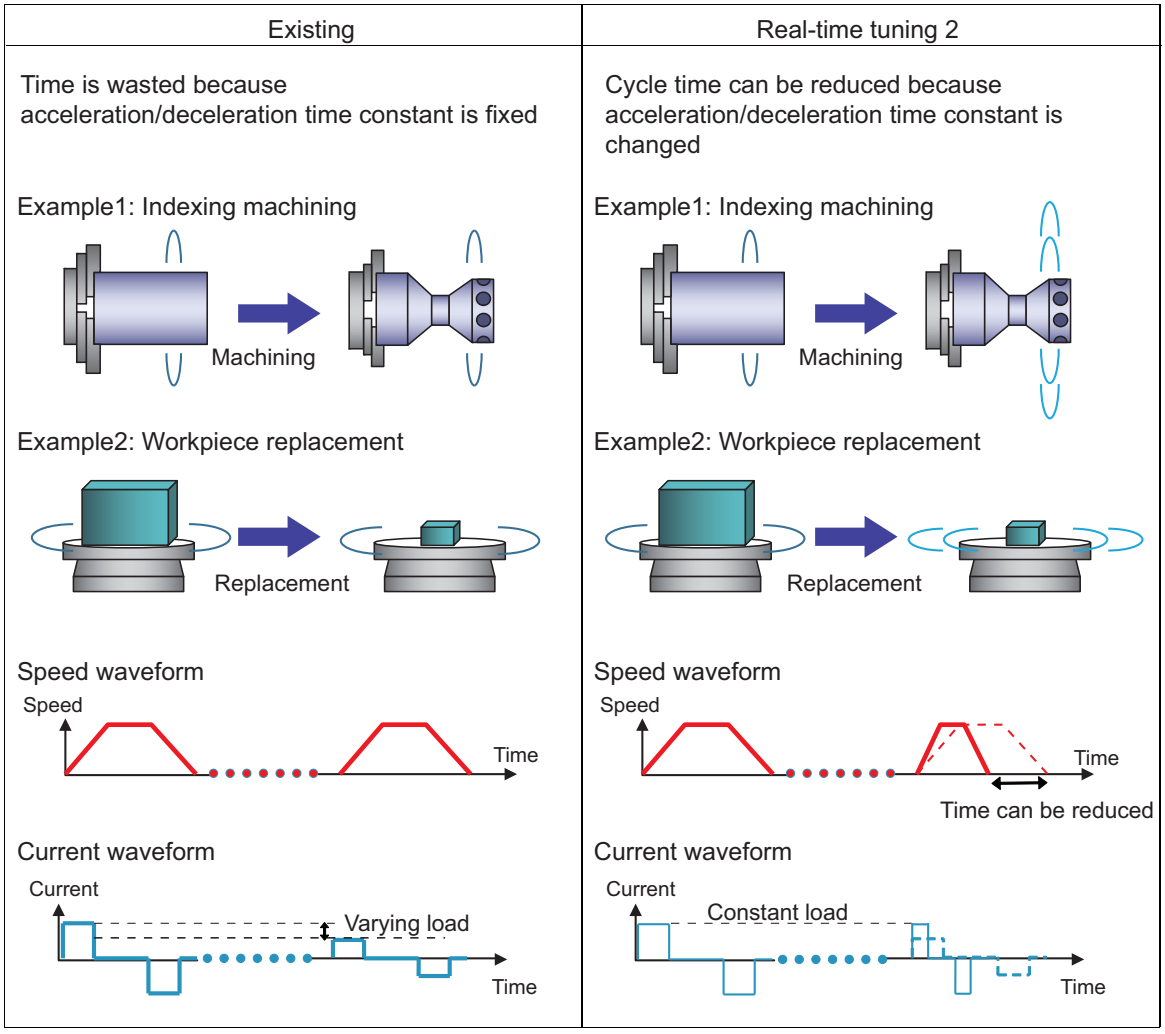
This function estimates the inertia (or workpiece weight) of mechanical system and changes the speed control gain automatically according to the estimation results to suppress mechanical vibration.
For example, this suppresses vibration caused when a large workpiece becomes smaller in turning machining as illustrated in Example 1 below. This function also suppresses vibration caused when a workpiece that is relatively small for the table is replaced by a larger one as illustrated in Example 2 below.
Using this function, users can expect suppression of vibration caused by inertia fluctuation, as well as reduction of machining time through adaptation of appropriate speed control gain.
This function also commands a stop of speed control gain change and displays the estimated inertia ratio and magnification of speed control gain (ratio of the changed speed control gain to the initially set gain).



17.2.20.2 Real-Time Tuning 2 (Rapid Traverse Time Constant)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	—
L	Δ	Δ	Δ	Δ	○	○	—

This function estimates the inertia of workpiece and optimizes the operation acceleration/deceleration time constant automatically according to the estimation results.
This allows the cycle time to be reduced when the inertia of workpiece fluctuates considerably.
This function is useful when the inertia of workpiece fluctuates considerably during machining as illustrated in Example 1: Indexing machining and Example 2: Workpiece replacement on the rotary table below.
In Example 1, the inertia of workpiece is reduced gradually as machining progresses. In Example 2, the inertia of workpiece on the rotary table is reduced by workpiece replacement. When the inertia of workpiece is large in Example 1 or Example 2, the machine tool is required to be operated at a low acceleration to suppress the vibration for the workpiece and overload for the motor. On the other hand, when the inertia of workpiece is smaller, the operation at a higher acceleration is available. In other words, acceleration/deceleration time constant can be reduced when the inertia of workpiece is smaller.
In the existing specifications, the operation acceleration/deceleration time constant is fixed regardless of the inertia of workpiece, which wastes time because a large time constant is applied even if the inertia of workpiece is small.
On the other hand, this function enables the operation with optimum acceleration/deceleration time constant according to the inertia of the workpiece. Therefore, machining is performed while reducing the acceleration/deceleration time constant in example 1 or example 2. This allows the tact time to be reduced.
Furthermore, the large inertia workpiece is not machined with small acceleration/deceleration time constant since the optimum time constant is adjusted automatically. This enables suppression of vibration and overload.



17.2.21 Constant Torque Control

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

■ Constant torque control

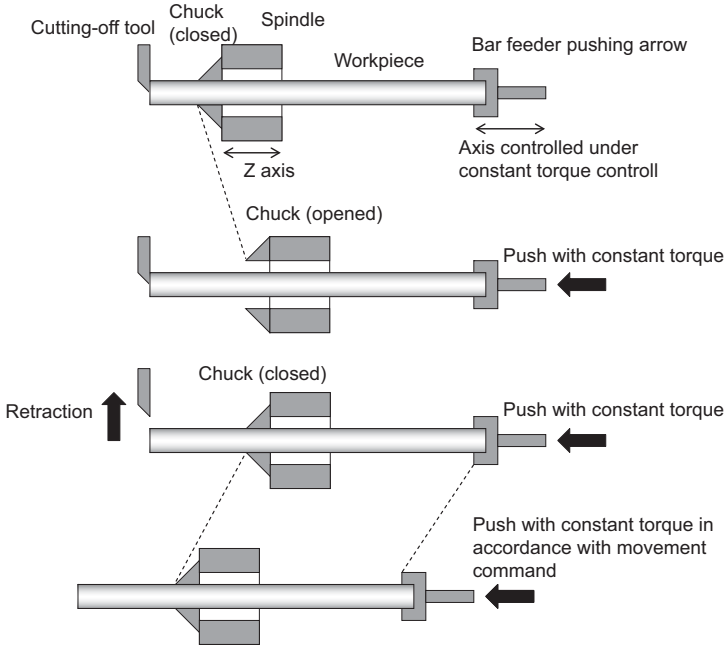
By setting "Constant torque control request axis" signal to ON from PLC, the servo motor of the designated axis outputs torque set by the parameter in a constant direction. By using this function for a servo motor which drives a bar feeder, the bar feeder can push the workpiece with constant torque regardless of whether it is during movement, stop, or acceleration/ deceleration.

Proportional torque stopper control is a function which enables a servo motor of the designated axis to generate torque set with the parameter in the pushing direction by setting "Proportional torque stopper control request axis" signal (R2620) to ON from PLC. When position droop is generated, the motor generates torque in proportion to the position droop in the pushing direction to keep the pushing position.

Note

(1) Do not cancel the constant torque control or the proportional torque stopper control for an axis moving under manual or automatic operation. Otherwise the fixed workpiece, fixed turret, etc., may get loose, and it is dangerous.

[Example of constant torque control]



■ Droop cancel under constant torque control

Droop cancel under constant torque control cancels the position droop generated during constant torque control without moving an axis.
Note that the droop cancel under constant torque control must be commanded while the axis is pushed against the workpiece, etc.

■ Proportional constant torque control

Proportional torque stopper control generates constant torque in the pushing direction according to the value designated to the parameter. When position droop is generated, the motor generates torque in proportion to the position droop in the pushing direction to keep the pushing position.

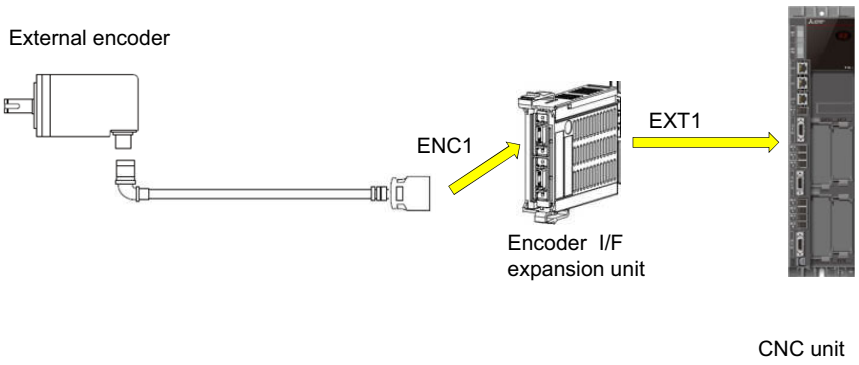
17.2.22 External Encoder Position Output I/F

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

Encoder I/F expansion unit is required for M800VW/M80VW.

This function outputs the position (angle) of external encoder to PLC device and CNC drive monitor screen based on the input pulses from the encoder.
This function can be used to monitor the position of angular head which the external encoder is connected to.
The position to output to PLC device and CNC drive monitor screen can be cleared to "0" with PLC signal.

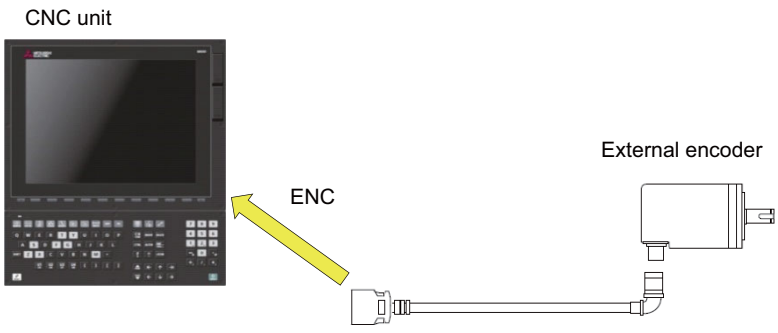
[M800VW/M80VW connection diagram]



Note

(1) Encoder I/F expansion unit can be used for ENC1 only.

[M800VS/M80V connection diagram]



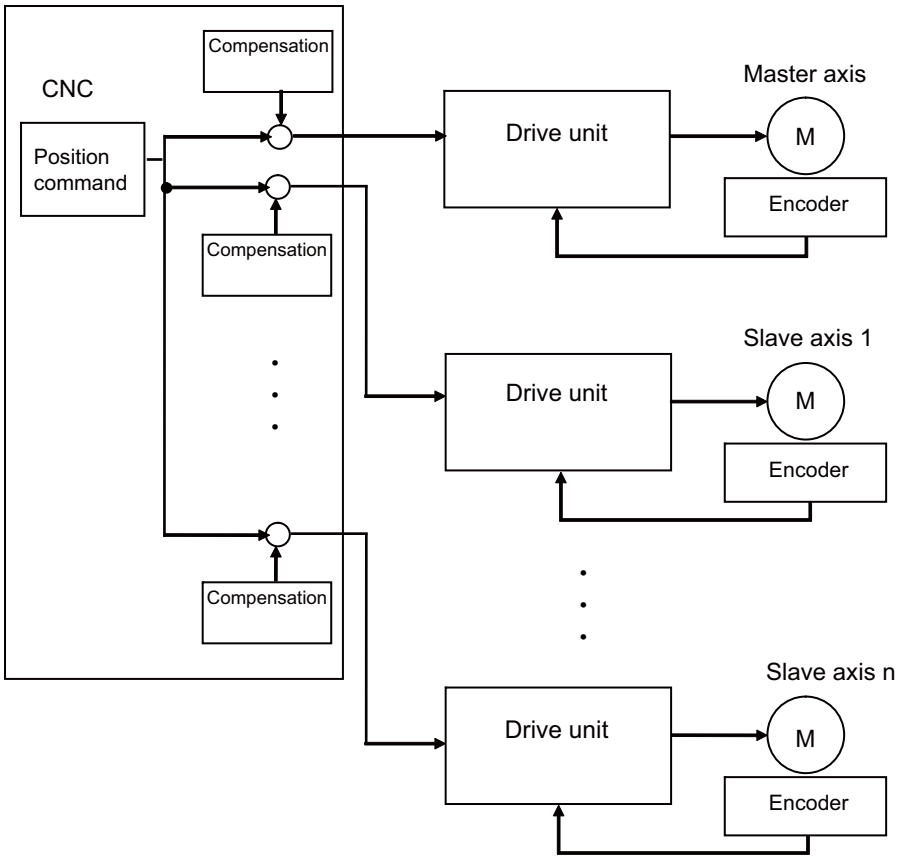
17.2.23 Multiple-Axis Synchronization Control

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	—	—	—	—	—	—	—

Multiple-axis synchronization control is the function that enables multiple slave axes to be synchronized with commands to the master axis. This function can be used, for example, in multi-head machine configurations.
The axis that serves as the reference of synchronization is called "master axis", whereas the axis that follows the master axis is called "slave axis".

- The synchronized axes are controlled with travel commands to the master axis.
- For one master axis, two or more slave axes can be set.
- Up to three combinations of master axis and slave axes can be set for all the part systems.

[Configuration example of multiple-axis synchronization control]



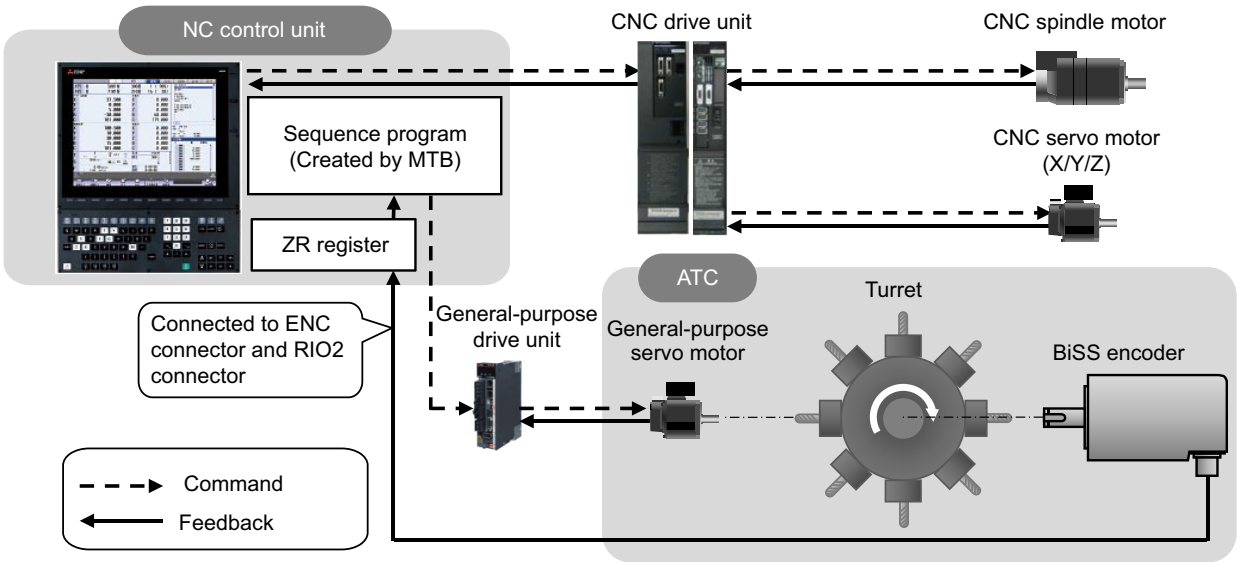
The "compensation" in the figure indicates the pitch error compensation, the thermal expansion compensation, the backlash compensation and the external machine coordinate compensation. Even under the multiple-axis synchronization control, the compensation is performed independently for each axis.

17.2.25 BiSS Encoder I/F

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	○	○	—	○	○
L	—	—	○	○	—	○	○

This function outputs input data from the encoder compatible with BiSS-C (hereinafter referred to BiSS encoder) to the PLC device (ZR register).
This function can be used to monitor the position of the turret rotary axis by connecting the BiSS encoder.

Data length and data bit arrangement of the BiSS encoder differ depending on the encoder specifications.
Create a sequence program according to the specifications of the data sent from the encoder to calculate the encoder angle.
The figure below is an example of connection structure for 3-axis tapping center.



Note

(1) This function supports both BiSS-C and BiSS-C (unidirectional), but does not support access to the register in the device on the slave side of the BiSS-C.

17.2.26 Tool Head Hot Swapping

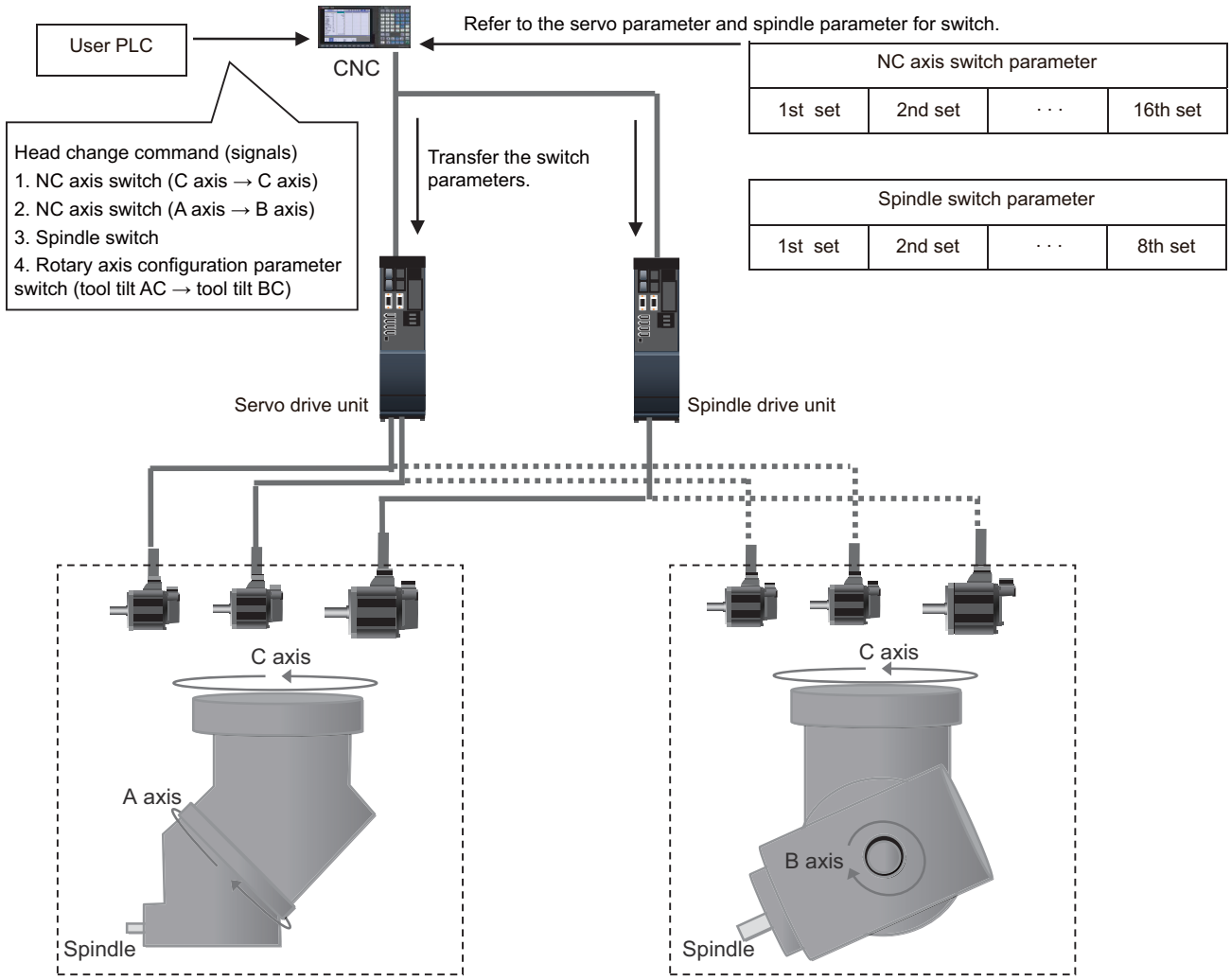
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	△	△	△	△	—	—	—
L	—	—	—	—	—	—	—

This is a function to perform machining by detaching the tool head connected to the machine temporarily and attaching another tool head.
When the tool head side includes the drive sections of the rotary axis and the spindle, the tool head can be changed without turning off the NC and the drive units. In that case, the servo parameter of the rotary axis and the spindle parameter of the spindle can be changed according to the tool head to be used. When the rotary axis configuration differs depending on the tool head, the rotary axis configuration parameter can be changed.

This function is used by commanding a combination of the following three types of PLC signals. The combination of the signals needs to be selected according to the specifications of the machine head.

- NC axis switch
- Spindle switch
- Rotary axis configuration parameter switch (available only for the first part system)

[Image of tool head change including two rotary axes and spindle]



17.3 PLC Operation

17.3.1 Arbitrary Feed in Manual Mode

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

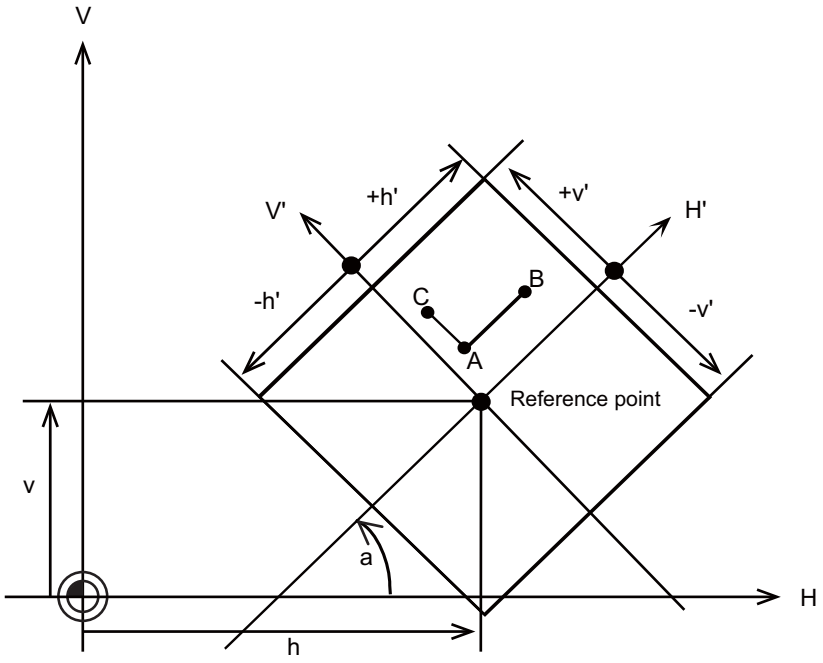
This is a function to control the NC control axis with command from the PLC in addition to the normal automatic operation command on the NC.
This function can be primarily used for axis control for ATC.
This function enables positioning of the axes from the PLC.

17.3.2 Circular Feed in Manual Mode

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

By specifying a hypothetical coordinate on the machine coordinate system from the user PLC, oblique linear interpolation or circular interpolation is executed with jog feed, handle feed, manual rapid traverse or incremental feed of the two axes designated by plane selection.
This function is valid only in the jog mode, handle mode, manual rapid traverse mode or incremental mode. This function cannot be used in the other manual modes and automatic operation modes.
This function operates for the horizontal axis (H axis) and vertical axes (V axis) determined by the parameters and PLC device settings. This function cannot be used for the other NC axes and PLC axis. When this function is valid, all the axes other than H axis and V axis move as usual.

■ Oblique linear interpolation



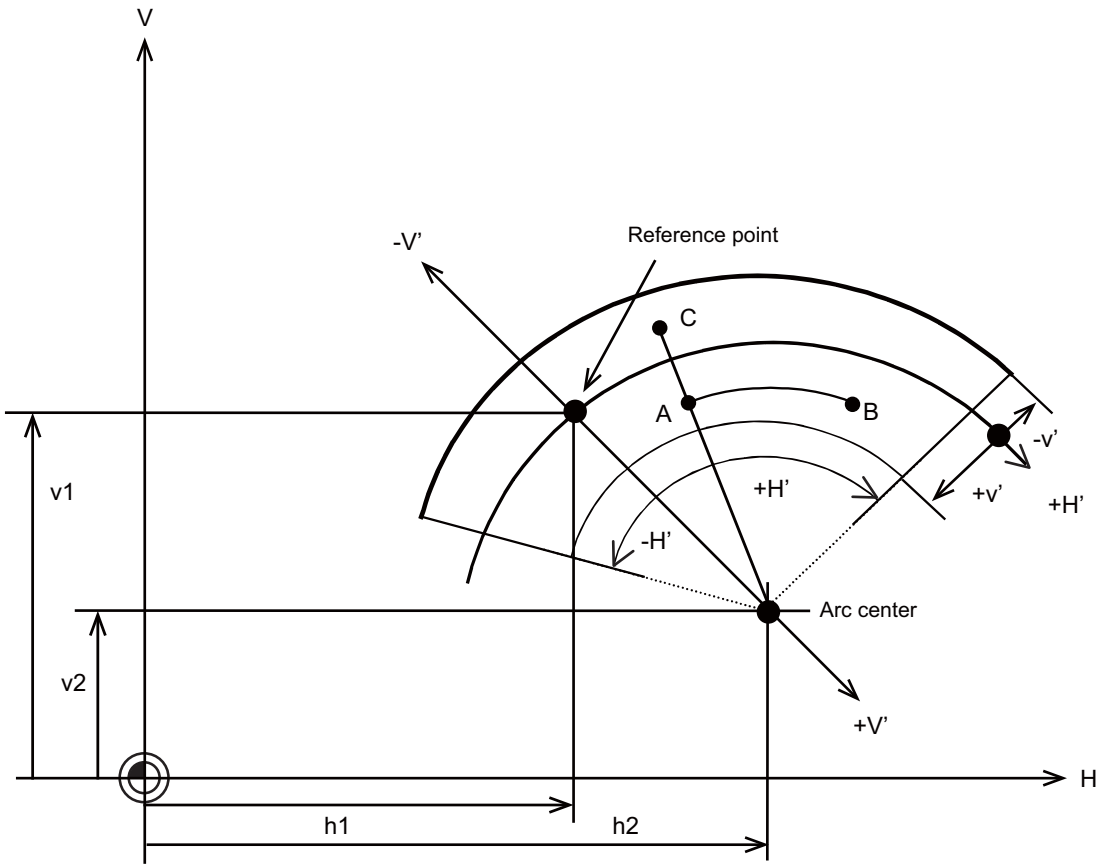
When the circular feed in manual mode (oblique linear interpolation) is valid:
- Jog/handle feed of H axis results in the movement in parallel with H' axis (see above).
- Jog/handle feed of V axis results in the movement in parallel with V' axis (see above).

Operation example:
- When handle feed is performed in the "+" direction of H axis at the current position (point "A"), the axis position moves to point "B". Following this operation, when handle feed is performed in the "-" direction of H axis at point "B", the axis position moves to point "A".
- When handle feed is performed in the "+" direction of V axis at the current position (point "A"), the axis position moves to point "C". Following this operation, when handle feed is performed in the "-" direction of V axis at point "C", the axis position moves to point "A".

These are set in the R registers.
a: Gradient angle
h, v: Reference point coordinate
±h': H' travel range
±v': V' travel range

■ Circular interpolation

By specifying a hypothetical coordinate on the machine coordinate as shown in the figure below, jog feed or handle feed can be executed on the hypothetical coordinate.



When the circular feed in manual mode (circular interpolation) is valid:
- Jog/handle feed of H axis results in the movement along the arc whose radius is the distance from the arc center to the current position.
- Jog/handle feed of V axis results in the movement on the line connecting the current position and the arc center.

Operation example:

- When handle feed is performed in the "+" direction of H axis at the current position (point "A"), the axis position moves to point "B". Following this operation, when handle feed is performed in the "-" direction of H axis at point "B", the axis position moves to point "A". The movement is done along the circumference of the arc whose radius is the distance from the arc center to the current position (point "A").
- When handle feed is performed in the "-" direction of V axis at the current position (point "A"), the axis position moves to point "C". Following this operation, when handle feed is performed in the "+" direction of V axis at point "C", the axis position moves to point "A".

These are set in the R registers.
h1, v1: Reference point coordinate
h2, v2: Arc center coordinate
±h': H' travel range
±v': V' travel range

17.3.3 PLC Axis Control

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

Over and above the NC control axes, this function enables axes to be controlled independently by commands based on the PLC.

[Specifications]

Item	Details
Number of control axes	Refer to " 1.1.2.3 Max. Number of PLC Axes".
Simultaneous control axes	The PLC control axis is controlled independently of the NC control axis. Simultaneous start of multiple PLC axes is possible.
Command unit	Least command increment 0.001 mm (0.0001 inch) 0.0001 mm (0.00001 inch) 0.00001 mm (0.000001 inch) 0.000001 mm (0.0000001 inch) (The command increment depends on the parameter setting.)
Feed rate	0 to 1000000 mm/min (0 to 100000 inch/min) (Speed is fixed regardless of the unit system.)
Movement commands	Incremental commands from current position Absolute commands for machine coordinate system 0 to ±99999999 (The command increment depends on the parameter setting.)
Operation modes	Rapid traverse, cutting feed Jog feed (+) (-) Reference position return feed (+) (-) Handle feed
Backlash compensation	Available
Stroke end	Devices can be allocated arbitrarily
Soft limit	Available
Rotary axis command	Available For absolute commands: Rotates by the amount within 1 rotation (Rotates the remainder of the rotation amount divided by rotational axis division count.) For incremental commands: Rotates by assigned amount
Inch/mm changeover	None Set to the command that corresponds to the feedback unit.
Position detector	Encoder (Absolute position can also be detected.)

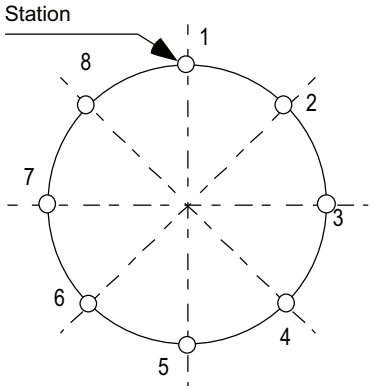
17.3.5 PLC Axis Indexing

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

PLC axis indexing allows a PLC axis to function as an auxiliary axis with no need for changing the user ladder used conventionally for an auxiliary axis. (The position of R register will be changed.)

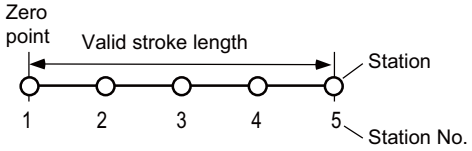
Command methods

Station method (for rotary axis) : One rotation (360°) of the rotary axis is equally divided to determine the stations (positioning destinations). The maximum number of divisions is 360.



Setting 8 stations (8 divisions)

Station method (for linear axis) : A valid stroke is equally divided to determine the stations (positioning destinations). The maximum number of stations is 360.



Setting 5 stations

The zero point is station 1, and the final end of the valid stroke is station 5.
When using a linear axis, the No. of equal divisions is "number of stations - 1".

Unequal station method : When the positioning stations are not at equal intervals, this method allows you to set up to 20 coordinate points arbitrarily to determine the station coordinates. This method is available for a rotary or linear axis.

Arbitrary coordinate designation method : This method allows you to transfer arbitrary coordinates (the absolute coordinates based on the zero point) from PLC to perform positioning.

Feed functions

Feed rate selection	:	Four different feeds per minute are set in the unit of " °/min" (rotary axis) or "mm/min" (linear axis). Which of the four is used is switched by the PLC I/F.
Acceleration/deceleration method	:	The constant-gradient acceleration/deceleration is automatically controlled. The linear or soft acceleration/deceleration is selectable.
Acceleration/deceleration pattern designation method	:	Four different acceleration/deceleration patterns are set to be selected by switching with PLC I/F.
Short-cut control	:	A least movement distance is automatically judged when a rotary axis is rotated. Also the rotation direction can be designated by the command.

Operation functions

The following operation modes are available. Send a command from PLC to change the operation mode.

Automatic operation mode	:	The axis is positioned at the station No. designated by the start signal. When the start signal has turned OFF before the positioning is completed, the axis is positioned at the nearest station position. The arbitrary position command operation is also available: the positioning can be carried out to any position other than a station by directly commanding the positioning coordinates from the PLC. (Arbitrary position command operation)
Manual operation mode	:	While the start signal is ON, the axis is rotated at a constant speed in the designated direction. When the start signal has turned OFF, the axis is positioned at the nearest station position.
JOG mode	:	While the start signal is ON, the axis is rotated at a constant speed in the designated direction.
Incremental feed mode	:	This mode is to feed the axis by the designated distance at each activation.
Manual handle mode	:	The axis is moved by using a pulse generator.
Reference position return mode	:	This mode is to position at the reference position. The reference position return by the dog switch is not available.

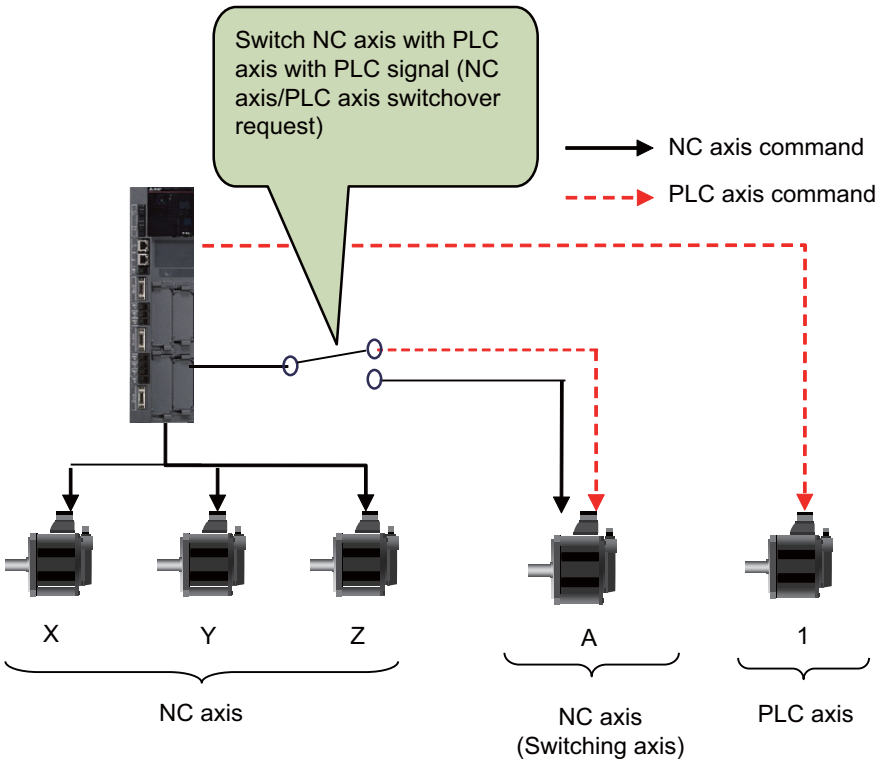
Operation support function

Position switch	:	A signal is output to the PLC interface when the machine has reached within the specified range.

17.3.6 NC Axis/PLC Axis Switchover

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	O	O	O
L	Δ	Δ	Δ	Δ	O	O	O

This function is to use one drive unit by switching the control from the NC or the PLC dynamically.
The function of the NC axis can be used during the NC axis control and the function of the PLC axis can be used during the PLC axis control respectively.
A part of function or signal of the NC axis can also be used during the PLC axis control.



17.4 PLC Interface

17.4.1 CNC Control Signal

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

Control commands to the CNC system are assigned from the PLC. Input signals with an A/D conversion function and skip inputs that respond at high speed can also be used.

■ Control signals

- Control signals for operations in automatic operation mode
- Control signals for operations in manual operation mode
- Control signals for program execution
- Control signals for interrupt operations
- Control signals for servo
- Control signals for spindle
- Control signals for mode selection
- Control signals for axis selection
- Control signals for feed rates

■ Analog voltage control [M system]

When an analog voltage is input to an external connector used to connect CNC analog inputs, the data corresponding to the input voltage can be read out in the prescribed file register. This data can be used for load meter displays, thermal deformation compensation, etc. (Maximum 4 points)

■ Skip signals

When signals are input to the skip input interface, they are processed by interrupt processing. This enables functions requiring a high response speed to be implemented. (Maximum 8 points)

For further details, refer to the PLC Interface Manual.

17.4.2 CNC Status Signal

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

The status signals are output from the CNC system. They can be utilized by referencing them from the PLC.
These signals can also be output as analog data by setting the data from the PLC in the R register.

■ Status output functions

(1) Controller operation ready

When the controller power is turned ON and the controller enters the operation ready status, the "Ready" signal is output to the machine.
Refer to the PLC Interface Manual for details of the sequences from when the controller power is supplied to when the controller ready status is entered.

(2) Servo ready completion

When the controller power is turned ON and the servo system enters the operation ready status, the "Servo ready completion" signal is output to the machine.
Refer to the PLC Interface Manual for details of the sequences from when the power is supplied to when the "Servo ready completion" signal is turned ON.

(3) In automatic operation

Generally, if the "cycle start" switch is turned ON in the automatic operation mode (memory, MDI), this signal is output until the reset state or emergency stop state is entered by the M02, M30 execution or the "Reset & rewind" input to the controller using the reset button.

(4) In cycle start

The signal that denotes that the controller is operating in the automatic mode is output from the time when the cycle start button is pressed in the memory or MDI mode and the cycle start status has been entered until the time when the automatic operation is terminated in the automatic operation pause status entered by the "feed hold" function, block completion stop entered by the block stop function or resetting.

(5) In automatic pause

An automatic operation pause occurs and this signal is output during automatic operation from when the automatic pause switch is pressed ON until the cycle start switch is pressed ON, or during automatic operation when the mode select switch is changed from the automatic mode to the manual mode.

(6) In rapid traverse

The "In rapid traverse" signal is output when the command now being executed is moving an axis by rapid traverse during automatic operation.

(7) In cutting feed

The "In cutting feed" signal is output when the command now being executed is moving an axis by cutting feed during automatic operation.

(8) In tapping

The "In tapping" signal is output when the command now being executed is in a tapping modal which means that one of the statuses below is entered during automatic operation.

- G84, G88 (fixed cycle: tapping cycle)
- G84.1, G88.1 (fixed cycle: reverse tapping cycle)
- G63 (tapping mode)

(9) In thread cutting

The "In thread cutting" signal is output when the command now being executed is moving an axis by thread cutting feed during automatic operation.

(10) In rewinding

The "In rewinding" signal is output when the "Reset & rewind" signal is input by M02/M30, etc., during memory operation and the program currently being executed is being indexed.
The rewinding time is short, so there may be cases when it cannot be confirmed with the sequence program (ladder).

(11) Axis selection output

The "Axis selection output" signal for each axis is output to the machine during machine axis movement.

- Automatic mode

The signal is output during the movement command for each axis. It is output until the machine stops during stop based on feed hold or block stop.

- Manual mode (including incremental feed)

The signal is output while the axis is moving from the time when the jog feed signal is turned ON until the time when it is turned OFF and the machine feed stops.

- Handle feed mode

The signal is output at all times when the axis selection input is on.

(12) Axis movement direction

This output signal denotes the direction of the axis now moving, and for each axis a "+" (plus) signal and a "-" (minus) signal are output respectively.

(13) Alarm

This signal indicates the various alarm statuses that arise during controller operation. It is divided into the following types and output.

- System errors
- Servo alarms
- Program errors
- Operation errors

(14) In resetting

The "Reset" signal is output during the reset process when the Reset & rewind command is input to the controller with the "reset" button on the setting and display unit is pressed or when the "Reset" signal is input from the machine operation panel, etc.
This signal will also be output when the controller READY status is OFF, when the Emergency stop signal is input or when a servo alarm is occurring, etc.

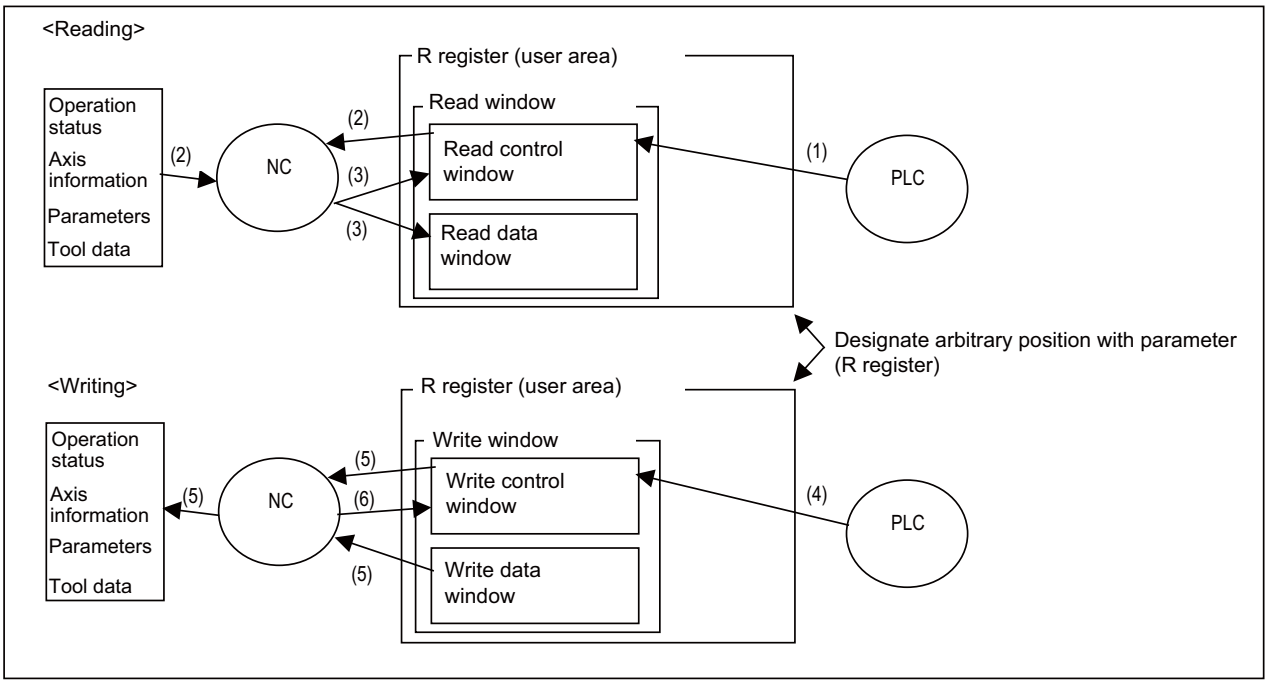
(15) Movement command completion

In the memory or MDI automatic operation, the "Movement command completion" signal is output when the command block in the machining program features a movement command and when that block command has been completed.
When the movement command and M, S, T or B command have been assigned in the same block, then the movement command signal can be used as a sync signal for either executing the processing of the M, S, T or B command at the same time as the command or executing it upon completion of the movement command.

17.4.3 PLC Window

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

Using the "read window" or "write window" that are arbitrarily allocated to the R register's user area, it is possible to read and write the CNC operation status, axis information, parameters and tool data etc.
The area used for the "read window" and "write window" has 1500 points for the backup area and 100 points for the non-backup area.
Up to three window areas can be specified simultaneously for both "read window" area and "write window" area. 16 R registers (18 R registers when an extended interface is used) are used for one read window or write window.
The read window is further divided into the "read control window" and "read data window". The write window is also divided into the "write control window" and "write data window".
Up to four data items can be successively read or written from the data designated in one read window or write window.



(1) PLC turns control signal ON.
(2) NC receives control signal, and reads designated data in control window.
(3) Results of NC read are set in read control window and read data window.
(4) PLC turns control signal ON.
(5) NC receives control signal, and writes contents of write data window corresponding to designated data in control window.
(6) Results of NC write are set in write control window.

17.4.4 External Search

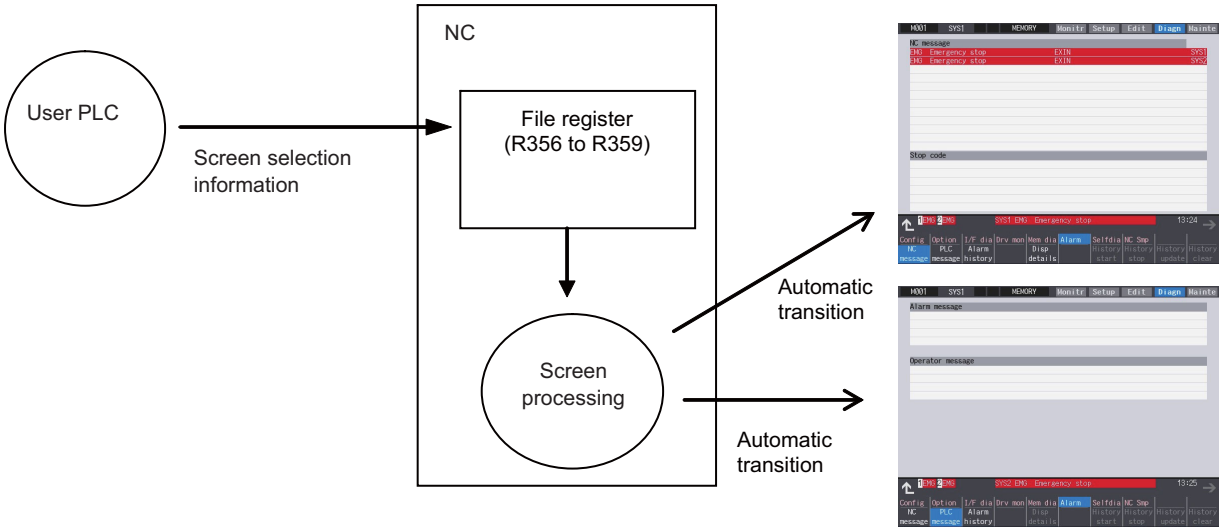
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

Machining program can be searched from user PLC by specifying the storage device of machining program, program No., sequence No. or block No.

17.4.5 Direct Screen Selection

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

This function enables automatic transition to an arbitrary screen by setting the screen selection information to the file registers from user PLC.
The screens that can be selected as the transition destination are Monitr, Setup, Edit, Diagn, Mainte and custom release screens.
This function is used to move to the custom screen being set by the MTB.

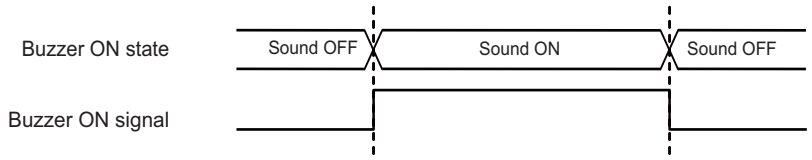


17.4.6 Buzzer Sound Control

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

This function gives a buzzer mounted on the NC keyboard by operating the PLC device.
The buzzer will be activated when the PLC device is turned ON, and be deactivated when the PLC device is turned OFF.
This is effective in applications of sounding a buzzer such as during the alarm occurrence, or for the program operation end notification.
This function also gives a buzzer when the operation sound disable has set.
This can be used when the NC keyboard is mounted.

[The buzzer sound image by the buzzer ON operation]



17.4.7 Extended ZR Device

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

Extended ZR device is an interface to access common variables and system variables #50000s from PLC program and GOT.
The range of extended ZR device is as follows.

Extended ZR device No.	Application
ZR140000 to 146419	GOT window function window setting command
ZR150000 to 155499	Access to system variables (#50000 to #52749)
ZR160000 to 169999	Access from PLC to common variables
ZR180000 to 199999	Access from GOT to common variables
ZR200000 to 207399	Access from PLC to common variables
ZR900000 to 914799	Access from GOT to common variables

17.5 Machine Contact I/O

17.5.1 Operation Panel I/O

17.5.1.1 DI:64/DO:64

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	—	—	□	—	—
L	□	□	—	—	□	—	—

17.5.1.2 DI:64/DO:64 + SDI:8

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	—	—	□	—	—
L	□	□	—	—	□	—	—

17.5.1.3 DI:96/DO:64

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	□	□	—	□	□
L	—	—	□	□	—	□	□

17.5.1.5 DI:64/DO:48/AO:1

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	□	□	—	□	□
L	—	—	□	□	—	□	□

17.5.2 Remote I/O

17.5.2.1 DI:32/DO:32

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	□	□	□	□	□
L	□	□	□	□	□	□	□

17.5.2.2 DI:64/DO:48

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	□	□	□	□	□
L	□	□	□	□	□	□	□

17.5.2.3 DI:64/DO:48+AO:1

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	□	□	□	□	□
L	□	□	□	□	□	□	□

17.5.2.5 DI:16/DO:8

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	□	□	□	□	□
L	□	□	□	□	□	□	□

17.5.2.6 DI:32/DO:32 + SDI:8/SDO:4

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	□	□	□	□	□
L	□	□	□	□	□	□	□

Functional safety expansion unit is required for M80V.

17.5.2.7 SDI:8/SDO:4

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	□	□	□	□	□
L	□	□	□	□	□	□	□

Functional safety expansion unit is required for M80V.

17.6 External PLC Link

17.6.1 CC-Link (Master/Local)

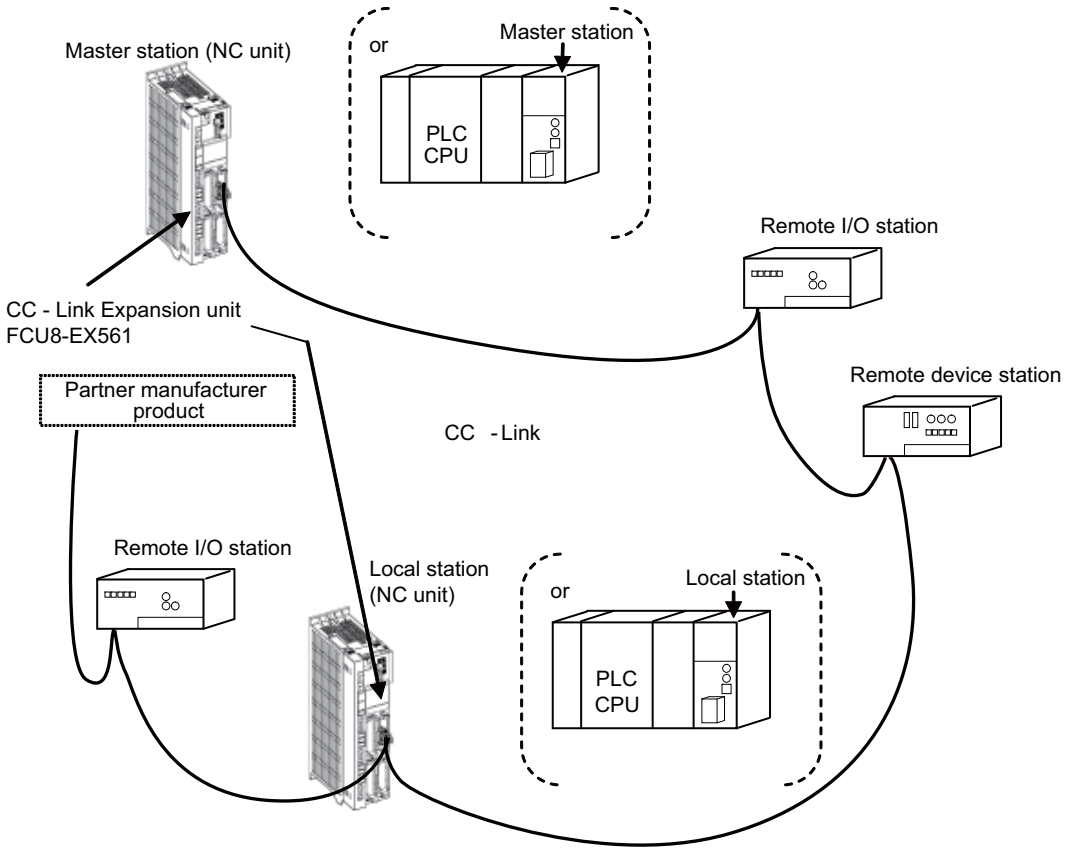
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	□	□	□	□	□
L	□	□	□	□	□	□	□

NC unit can be directly connected to the network to serve as the master/local station of the MELSEC CC-Link.
CC-Link unit is required for this connection.

■ Outline of CC-Link

- Distributing and installing each unit to the equipments such as conveyor line and mechanical device can simplify the wiring of the whole system.
- The ON/OFF data and numerical data such as input/output treated by each unit can be communicated easily and at high speed.
- The simple distribution system can be established by connecting several PLC CPUs or NCs.
- Connecting the device equipments made by the partner manufacturer can flexibly support various systems.

■ Outline drawing (M800VW Series)



Master station	This station controls the remote station and local station. One master station is required for one system.
Local station	This station contains the CPU and can communicate with the master and the other local stations.
Remote I/O station	Remote station that handles only bit information.
Remote device station	Remote station that handles bit information and word information.
Intelligent device station	This station allows the transient transmission. (Including local station)

■ Performance specifications

Mitsubishi Electric CNC M8V Series is compliant with CC-Link Ver. 2.00.

[CC-Link Ver. 2.00 specification]

Item		CC-Link master/local unit				
Maximum number of link points per system (*1)		Remote input/output (RX, RY)	8192 points each			
		Remote register (RWw) (Master St. -> Remote/Local St.)	2048 points			
		Remote register (RWr) (Remote/Local St. -> Master St.)	2048 points			
Extended cyclic setting			1 time setting	2 times setting	4 times setting	8 times setting
Number of link points per remote/local station	1 station occupied	Remote input/output (RX, RY)	32 points each	32 points each	64 points each	128 points each
		Remote register (RWw) (Master St. -> Remote/Local St.)	4 words	8 words	16 words	32 words
		Remote register (RWr) (Remote/Local St. -> Master St.)	4 words	8 words	16 words	32 words
	2 stations occupied	Remote input/output (RX, RY)	64 points each	96 points each	192 points each	384 points each
		Remote register (RWw) (Master St. -> Remote/Local St.)	8 words	16 words	32 words	64 words
		Remote register (RWr) (Remote/Local St. -> Master St.)	8 words	16 words	32 words	64 words
	3 stations occupied	Remote input/output (RX, RY)	96 points each	160 points each	320 points each	640 points each
		Remote register (RWw) (Master St. -> Remote/Local St.)	12 words	24 words	48 words	96 words
		Remote register (RWr) (Remote/Local St. -> Master St.)	12 words	24 words	48 words	96 words
	4 stations occupied	Remote input/output (RX, RY)	128 points each	224 points each	448 points each	896 points each
		Remote register (RWw) (Master St. -> Remote/Local St.)	16 words	32 words	64 words	128 words
		Remote register (RWr) (Remote/Local St. -> Master St.)	16 words	32 words	64 words	128 words
Number of occupied stations (Number of local stations)		Station 1 to station 4 (The station No. can be switched with parameters.)				
Baud rates		156 kbps/625 kbps/2.5 Mbps/5 Mbps/10 Mbps can be selected. (Baud rate can be selected with parameters.)				
Communication method		Polling method				
Synchronization method		Flame synchronization method				
Encode method		NRZI method				
Transmission path		Bus (conforming to EIA RS485)				
Transmission format		HDLC compliant				
Error control system		CRC ($X^{16} + X^{12} + X^5 + 1$)				

<table>
<tr><th>Item</th><th>CC-Link master/local unit</th></tr>
<tr><td>Maximum number of connected stations</td><td>64 units. Note that the following two conditions must be satisfied.
Condition 1:
• (a + a2 + a4 + a8) + (b + b2 + b4 + b8) × 2 + (c + c2 + c4 + c8) × 3 + (d + d2 + d4 + d8) × 4 ≤ 64
• (a × 32 + a2 × 32 + a4 × 64 + a8 × 128) + (b × 64 + b2 × 96 + b4 × 192 + b8 × 384) + (c × 96 + c2 × 160 + c4 × 320 + c8 × 640) + (d × 128 + d2 × 224 + d4 × 448 + d8 × 896) ≤ 8192
• (a × 4 + a2 × 8 + a4 × 16 + a8 × 32) + (b × 8 + b2 × 16 + b4 × 32 + b8 × 64) + (c × 12 + c2 × 24 + c4 × 48 + c8 × 96) + (d × 16 + d2 × 32 + d4 × 64 + d8 × 128) ≤ 2048
a: Number of units when 1 station is occupied in 1 time extended cyclic setting
b: Number of units when 2 stations are occupied in 1 time extended cyclic setting
c: Number of units when 3 stations are occupied in 1 time extended cyclic setting
d: Number of units when 4 stations are occupied in 1 time extended cyclic setting
a2: Number of units when 1 station is occupied in 2 times extended cyclic setting
b2: Number of units when 2 stations are occupied in 2 times extended cyclic setting
c2: Number of units when 3 stations are occupied in 2 times extended cyclic setting
d2: Number of units when 4 stations are occupied in 2 times extended cyclic setting
a4: Number of units when 1 station is occupied in 4 times extended cyclic setting
b4: Number of units when 2 stations are occupied in 4 times extended cyclic setting
c4: Number of units when 3 stations are occupied in 4 times extended cyclic setting
d4: Number of units when 4 stations are occupied in 4 times extended cyclic setting
a8: Number of units when 1 station is occupied in 8 times extended cyclic setting
b8: Number of units when 2 stations are occupied in 8 times extended cyclic setting
c8: Number of units when 3 stations are occupied in 8 times extended cyclic setting
d8: Number of units when 4 stations are occupied in 8 times extended cyclic setting
Condition 2:
• {(16 × A) + (54 × B) + (88 × C)} ≤ 2304
• A: Number of remote I/O stations ≤ 64 units
• B: Number of remote device stations ≤ 42 units
• C: Number of local stations, standby master stations and intelligent device stations ≤ 26 units</td></tr>
<tr><td>Remote station No.</td><td>1 to 64</td></tr>
<tr><td>Maximum total cable length and each cable length between stations</td><td>CC-Link Ver.1.10 compliant cable (When a 110 Ω of terminating resistance is used.) (*2)
<table>
<tr><th>Baud rates</th><th>Cable length between stations</th><th>Maximum total cable length</th></tr>
<tr><td>156 kbps</td><td rowspan="5">20 cm or more</td><td>1200 m</td></tr>
<tr><td>625 kbps</td><td>900 m</td></tr>
<tr><td>2.5 Mbps</td><td>400 m</td></tr>
<tr><td>5 Mbps</td><td>160 m</td></tr>
<tr><td>10 Mbps</td><td>100 m</td></tr>
</table></td></tr>
<tr><td>Connection cable</td><td>CC-Link Ver.1.10 compliant cable (3-core twisted pair cable with shield) (*3)</td></tr>
<tr><td>RAS function(*4)</td><td>• Automatic return function
• Slave station cut-off function
• Error detection by the link special relay/register</td></tr>
</table>

(*1) If the points which can be reserved as the device for the CC-Link in the NC side does not reach 8192 points, the number of points which can be reserved in the NC side is the maximum number of link points per system. This applies for the remote register (RWw, RWr), as well.

(*2) When CC-Link Ver.1.00 compliant cables are mixed, the cable length between stations and the maximum total length of the cable should follow the specifications for CC-Link Ver.1.00.

(*3) This can be used with the cables made by different manufacturers as long as they are Ver.1.10 compliant cables.

(*4) RAS stands for "Reliability", "Availability" and "Serviceability".

■ Usable functions

Of all the CC-Link functions, the following functions can be used with the NC.

○: Available

×: Not available

-: Not relevant

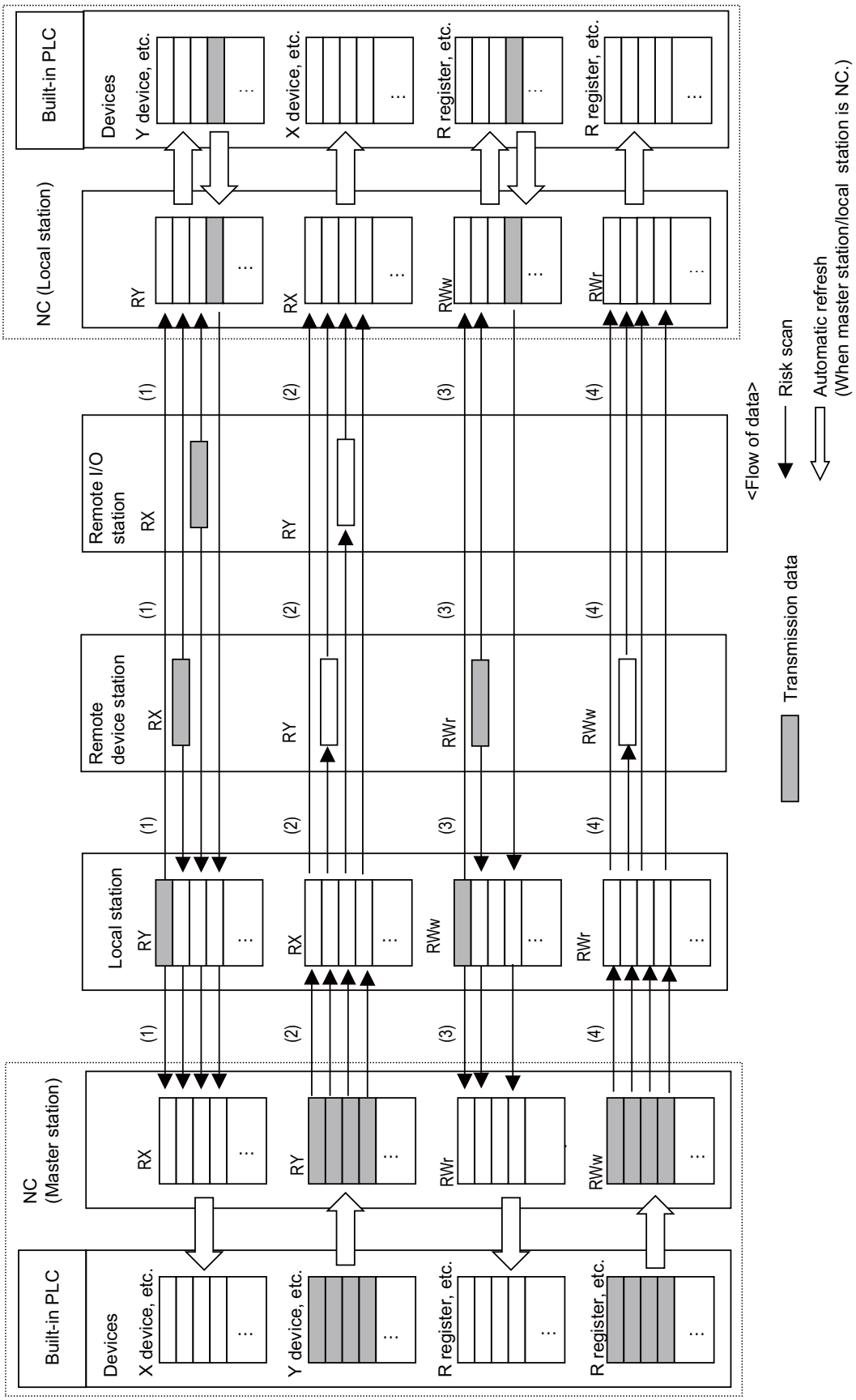
	Function item		MELSEC		NC unit	
			Master station	Local station	Master station	Local station
Method	Ver.1.00		○	○	○	○
	Ver.2.00		○	○	○	○
Master function	Communication between master station and remote I/O station		○	-	○	-
	Communication between master station and remote device station		○	-	○	-
	Communication between master station and local station		○	○	○	○
	Mixed system communication		○	○	○	○
	Reserved station function		○	-	○	-
	Error cancel station function		○	-	○	-
	Setting of data link status when trouble occurs in CPU of master station		○	○	○	○
	Registration of parameters in E2PROM		○	-	○ (*2)	-
	Setting of input data status from data link trouble station		○	○	○	○
	Unit resetting by sequence program		○	○	○	○
	Data link stop/restart		○	○	○	○
	Parameter registration function		○	○	○ (*2)	○ (*2)
	Automatic refresh function		○	○	○	○
	Scan synchronization function	Synchronous mode	○	-	○	-
		Asynchronous mode	○	○	○	○
Setting and display function	LED diagnosis status		16-point display (A1SJ61QBT11)		16-point display	
	Station number setting		Unit front panel switches		Parameters(*2)	
	Baud rate setting					
	Mode setting switch					
	Condition setting				H/W switches	
RAS function	Automatic return function		○	○	○	○
	Slave station cut-off function		○	-	○	○
	Data link status check (SB/SW)		○	○	○	○
	Off-line test		○	○	○	○
	On-line test		○	○	○	○
	Monitor diagnosis		○	○	×	×
	Standby master function(*3)		○	-	○	-
	Temporary error cancel station designation function		○	-	○	-
Exclusive instruction	READ instruction / SREAD instruction(*1)		○	○	○	○
	WRITE instruction / SWRITE instruction(*1)		○	○	○	○
	RIRD instruction / RIWT instruction(*1)		○	○	×	×

(*1) The transient instruction cannot be used.

(*2) The parameter for the CC-Link is set with the GX Developer.

(*3) With this function, when an error occurs in the master station, switch to the standby master station so that data link can be continued.

◆ Communication data flow
The flow of data communicated by the CC-Link's link scan is as follows. (The master station and local station of MELSEC CPU can be also mixed.)



- By executing a link scan, data in the remote I/O station and remote device station's remote input (RX) and in the local station's remote output (RY) is transmitted to the master station's remote input (RX) and the local station's remote output (RY).
- By executing a link scan, data in the master station's remote output (RY) is transmitted to the remote I/O station and remote device station's remote output (RY) and the local station's remote input (RX).
- By executing a link scan, data in the remote device station's remote register (RWr) and the local station's remote register (RWw) is transmitted to the master station's remote register (RWr) and the local station's remote register (RWw).
- By executing a link scan, data in the master station's remote register (RWw) is transmitted to the remote device station's remote register (RWw) and the local station's remote register (RWr).

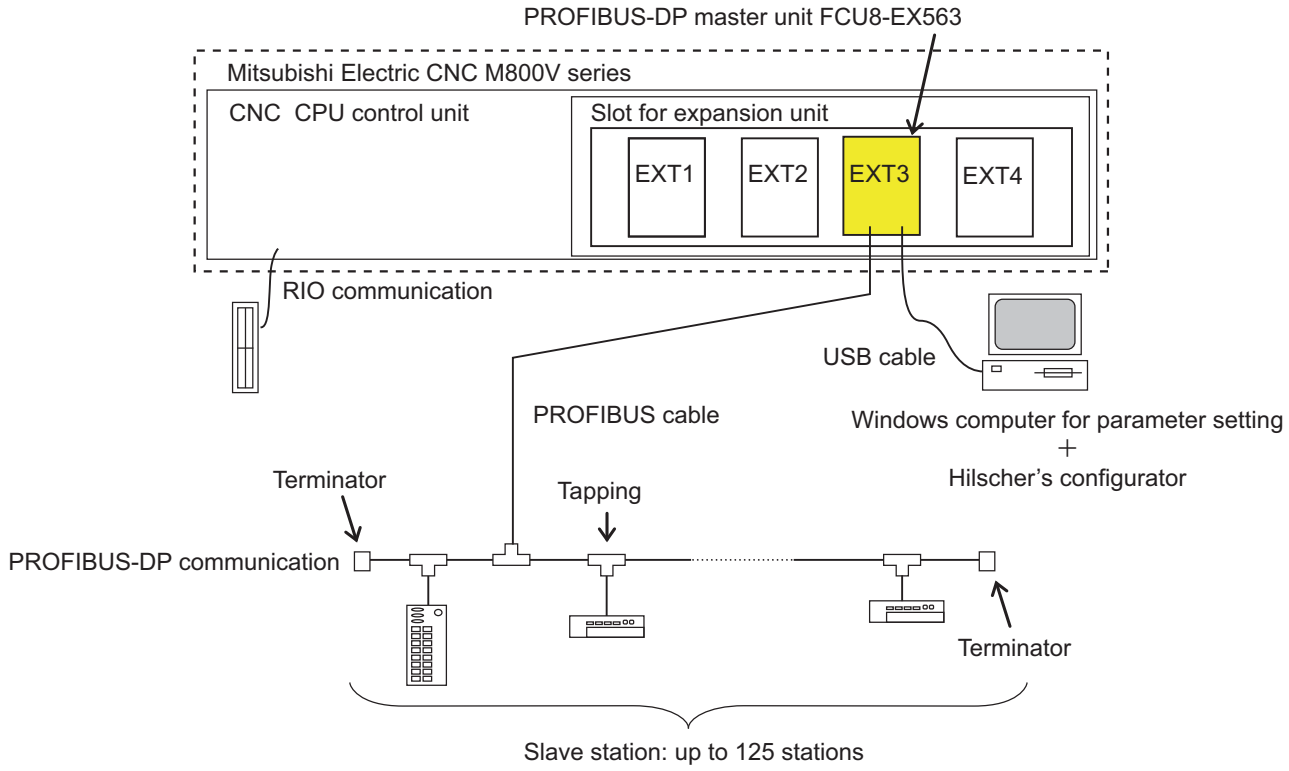
17.6.2 PROFIBUS-DP (Master)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	□	□	□	□	□
L	□	□	□	□	□	□	□

On Mitsubishi Electric CNC M8V Series, by mounting the PROFIBUS-DP master unit on the expansion slot or option relay unit and by connecting the unit with the slave stations compatible with PROFIBUS-DP communication, NC can input/output the devices to/from the slave stations as the master station for PROFIBUS-DP communication. CNC can be connected to up to 125 slave stations.
For the I/O data communication function (Cyclic communication), the maximum number of device I/O points is 8192 each. The CNC handles all the devices input/output to/from the PLC as bit device data. At this time, the maximum number of device I/O points of CNC's remote I/O is 768 each.
Using the aperiodic data communication function (Acyclic communication) with the slave station, data exchange can be performed to the slave station via PLC word device. Aperiodic data communication function can be performed with the I/O data communication function simultaneously.
Also, an error that occurred in the slave station can be acquired or responded via PLC word device as diagnosis data or alarm information.
There are two types of PLC I/O device assignment method for PROFIBUS-DP, fixed or arbitrary assignment, which you can select with the parameter. According to the assignment method, the number of available device points differs as follows:

	Fixed device assignment	Arbitrary device assignment
Maximum number of input device points	512 bits	8192 bits
Maximum number of output device points	512 bits	8192 bits

Hilscher's Fieldbus communication control circuit (COMX module) is mounted on PROFIBUS-DP master unit and the parameter setting is carried out using Hilscher's configurator "SYCON.net".
However, two PROFIBUS-DP master units cannot be mounted at a time. If these are mounted at the same time, both units stop their operations and the error occurs.



17.6.3 CC-Link IE Field (Master/Local)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	□	□	□	□	□
L	□	□	□	□	□	□	□

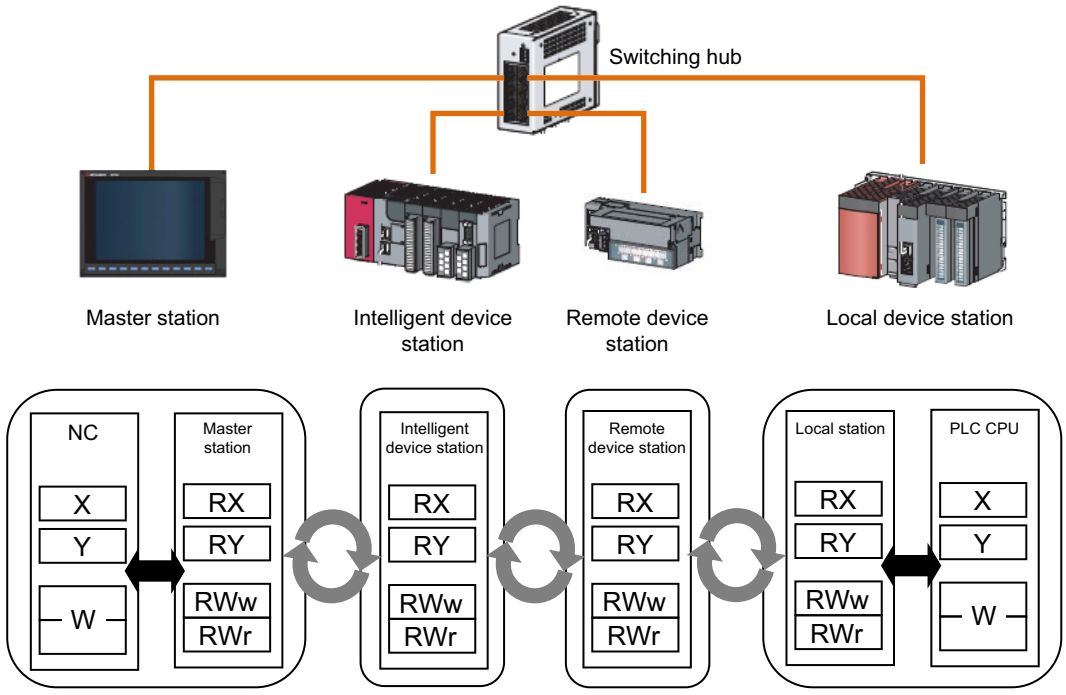
CC-Link IE Field is high-speed and large capacity open field network using Ethernet (1000BASE-T).

By mounting CC-Link IE Field expansion unit on the expansion slot of NC control unit, NC can be operated as master or local station of CC-Link IE Field. By connecting CC-Link IE Field expansion unit with CC-Link IE Field unit, data communication with the connected unit can be performed.
NC can be set as master or local station, but cannot be set as sub master station.
When CC-Link IE Field expansion unit is not mounted on the expansion slot, CC-Link IE Field cannot be used. CC-Link IE Field related parameter screen is not displayed on NC screen, either.

■ Data communication

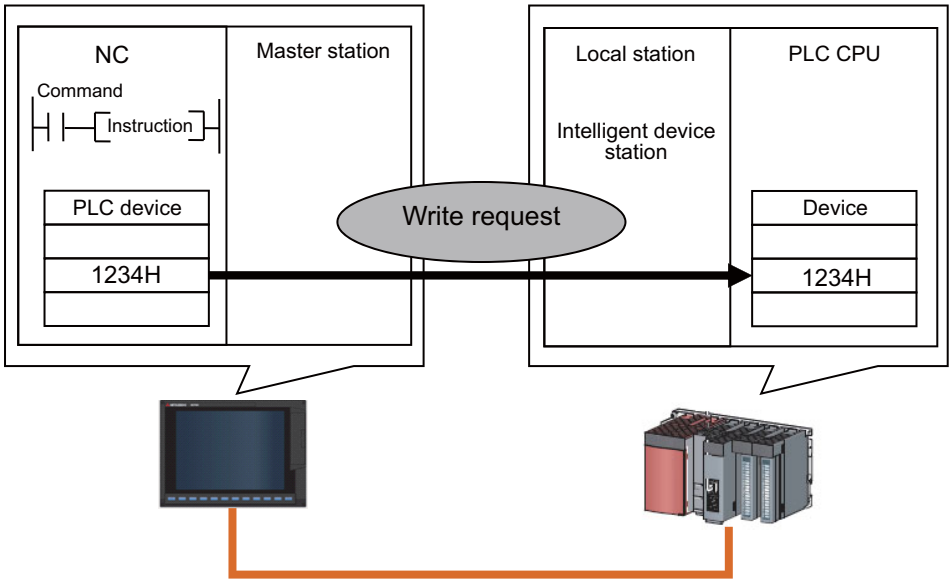
High-speed and large-capacity data communication is available between a master station and device stations on CC-Link IE Field Network. The network supports cyclic transmission (periodic communication) and transient transmission (non-periodic communication).

[Cyclic transmission (periodic communication)]
Data communications can be made periodically between stations of the same network.



[Transient transmission (non-periodic communication)]

Data are transmitted upon request.



■ 1 Gbps communication speed

1 Gbps communication speed allows high-speed communication.

■ Use of Ethernet cable

A 1000BASE-T-compliant Ethernet is used for the connection interface. The wiring cost can be reduced because 1000BASE-T-compliant Ethernet cables are commercially available.

■ Flexible wiring for system arrangements

The network can be wired into star topology, line topology and ring topology.
For star topology, a 1000BASE-T-compliant switching hub can be used.
Wiring is highly flexible because a network can consist of a combination of star and line topologies. For example, the machine tools can be connected through a line topology whereas star and line topologies can be used within each machine.

17.6.5 EtherNet/IP

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	□	□	□	□	□
L	□	□	□	□	□	□	□

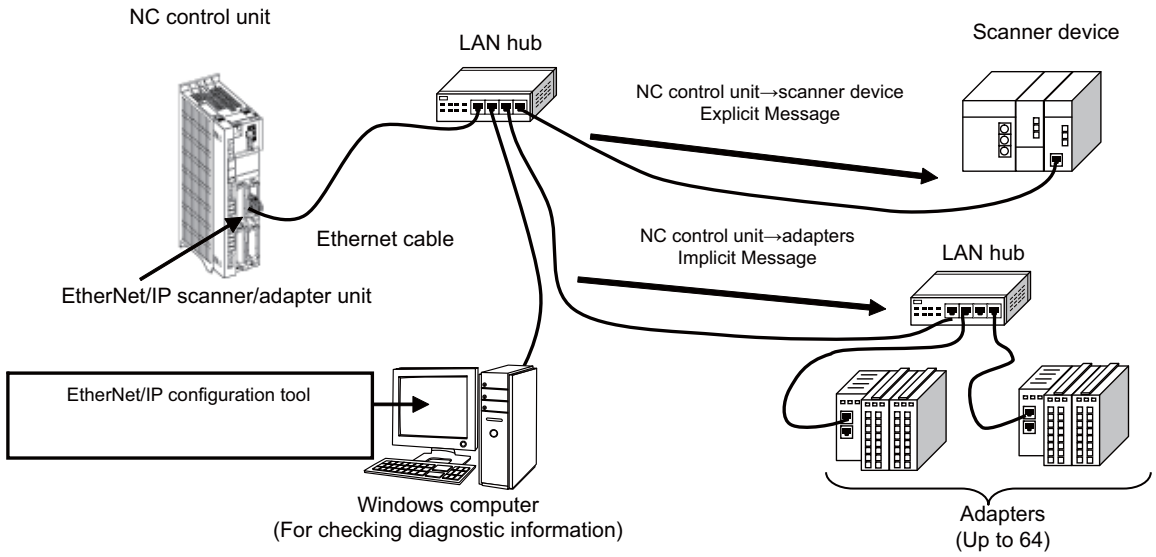
EtherNet/IP is the Ethernet for industry which ODVA (Open DeviceNet Vendor Association) suggests.
It communicates using TCP/IP which is the standard Ethernet technology and the communication protocol which is called CIP (Common Industrial Protocol). EtherNet/IP can be used as not only field network but also the network between controllers. Also, as the standard Ethernet technology is used as lower protocol at the same time, it can be used mixing various general purpose Ethernet devices.

By mounting EtherNet/IP scanner/adapter unit on the expansion slot or option relay unit of NC control unit to connect EtherNet/IP-compatible equipment, PLC devices can be input/output using NC control unit as a scanner or an adapter for EtherNet/IP communication.
When EtherNet/IP scanner/adapter unit is not mounted on the expansion slot or option relay unit, EtherNet/IP cannot be used. Ethernet/IP related parameter screen is not displayed, either.
Two EtherNet/IP scanner/adapter units cannot be mounted on the expansion slot or option relay unit. When two of them are mounted at the same time, both units stop operating and an alarm is output.

EtherNet/IP has two communication functions: Implicit message communication which communicates at regular intervals and Explicit message communication which communicates at arbitrary timing.

Message type	Outline
Explicit message	Communication based on TCP/IP. Communication method which sends and receives data at 1:1 at arbitrary timing. (Example) Parameter communication, reading diagnostic data, etc.
Implicit message	Communication based on UDP/IP. Communication method which sends and receives data at regular intervals (Cyclic). (Example) Transferring periodic data

[Connection diagram]



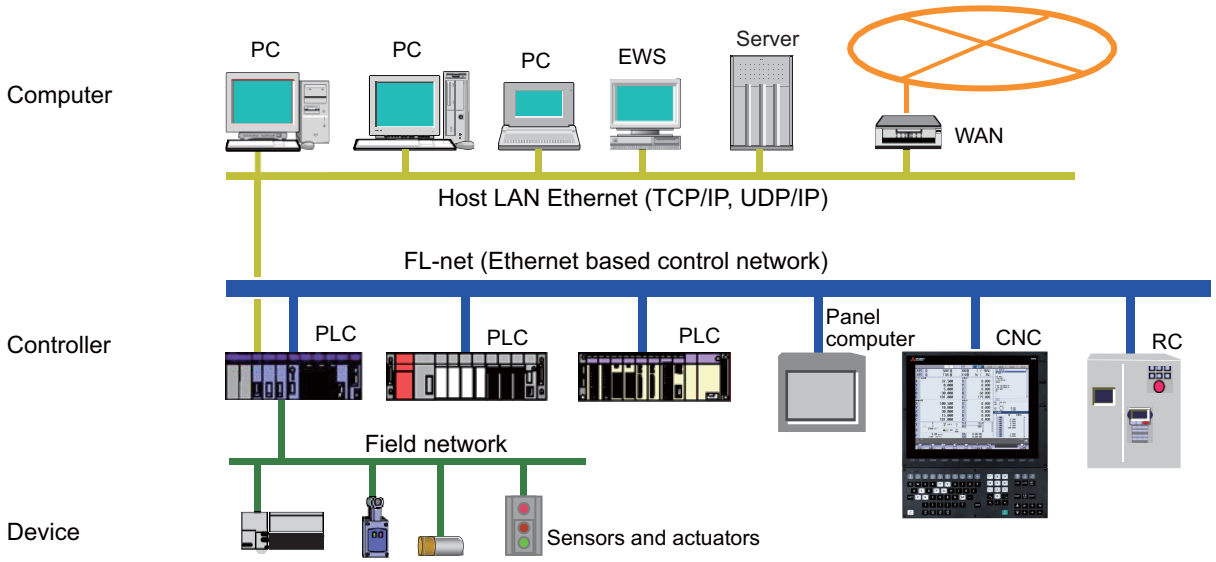
17.6.8 FL-Net

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	□	□	□	□	□
L	□	□	□	□	□	□	□

FL-net is a network capable of interconnecting various FA controllers such as the programmable controllers (PLC) and the CNC from many manufacturers, and personal computers for control and monitoring.
FL-net communication is made available by mounting an FL-net expansion unit on the expansion slot or option relay unit of NC control unit.

FL-net expansion unit of the NC control unit supports the following FL-net (OPCN-2) version.

FL-net version	Class
FL-net Version 3.01	Class 1 • Controller level network (Version 2.00 of FL-net) is available.



17.6.10 CC-Link IE Field Basic

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

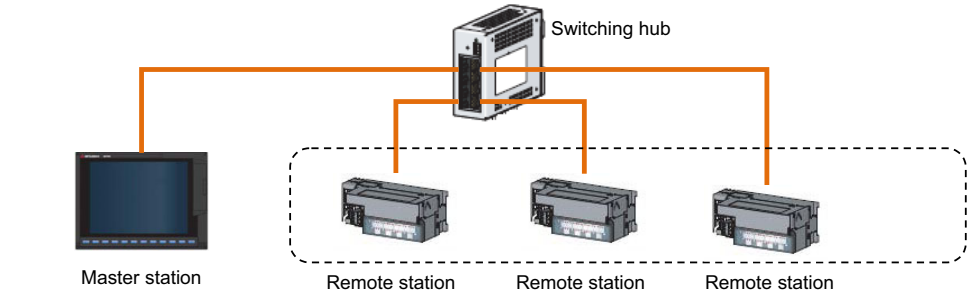
CC-Link IE field Basic is an FA network taking advantage of general Ethernet. This function is used to perform cyclic transmission between the master station and remote station using link devices.

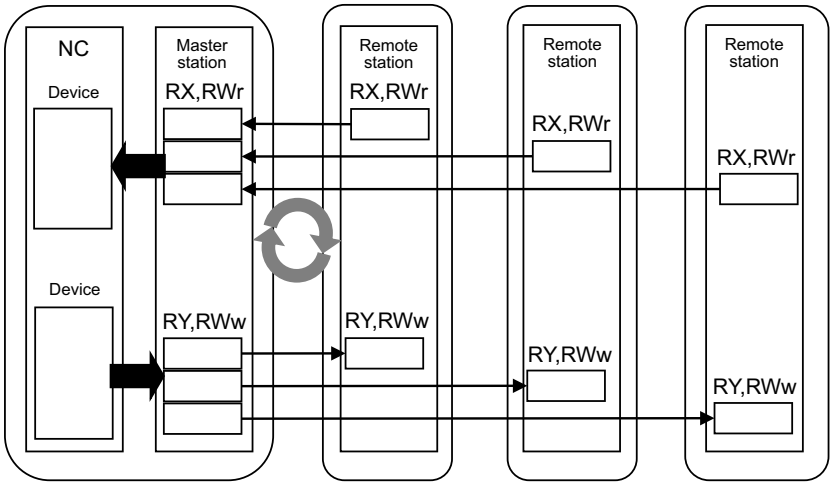
The CNC enables periodic (cyclic transmission) data communication between the master station and remote station by connecting the machine to the device conforming to CC-Link IE field Basic communication through the built-in Ethernet port. The data communicated is assigned to built-in PLC devices and can be referred to from the ladder program.
The CNC can function as a master station or as a remote station of CC-Link IE field Basic communication. Whether to use the CNC as the master station or as the remote station is determined by the parameters. The function as the master station or as the remote station cannot be used concurrently.
The parameters are used to set CC-Link IE field Basic communication, to specify the communication device, and to assign the link refresh devices.

The advantages of using CC-Link IE field Basic are described below.

■ Cyclic transmission (periodic communication)

Data communications can be made periodically between stations of the same network.





■ Reduction in system costs

The NC communicates through its built-in Ethernet port. This eliminates the need for a dedicated network unit, and therefore contributes to reduction in system costs.

■ Mixture of protocols

CC-Link IE field Basic can be used concurrently with other general Ethernet protocols. Therefore, the network can be configured using FTP, HTTP, SLMP, etc. However, a link scan time for cyclic transmission may increase due to those Ethernet protocols.

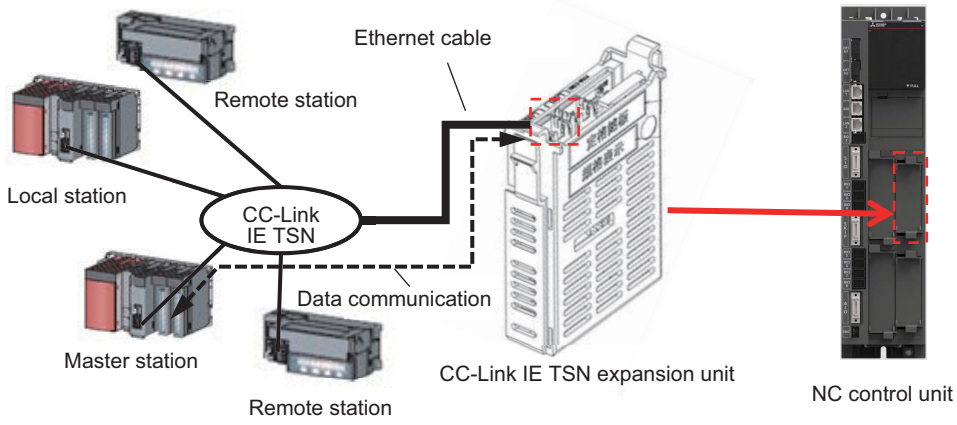
■ Easy-to-configure system

All the settings of CC-Link IE field Basic are made on the master station. Cyclic transmission is enabled by setting the IP address and the subnet mask on the remote station. There is no need for setting the information of the master station on the remote station.
In addition, there is no restriction on the connection order nor connecting locations of the devices as they are connected via a star network. It is therefore easy to connect an additional remote device.

17.6.11 CC-Link IE TSN (Remote Station)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	□	□	□	□	□
L	□	□	□	□	□	□	□

This function is used for data communication between a CC-Link IE TSN master device and a PLC device of the NC control unit through a CC-Link IE TSN expansion unit mounted on the NC.
The CC-Link IE TSN (Ver.2.0 Class A) remote station function can be added by mounting the CC-Link IE TSN expansion unit on an expansion slot of the NC control unit.
When the CC-Link IE TSN expansion unit is not mounted on the expansion slot of the NC control unit, the CC-Link IE TSN remote station function cannot be used. The parameter screen related to CC-Link IE TSN expansion unit is not displayed on the NC screen.



17.7 Installing S/W for Machine Tools

17.7.1 Customization (NC Designer2)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

Customization is a function which allows to display a user-created screen or user-created window as an HMI screen or as a screen of another application.

Note

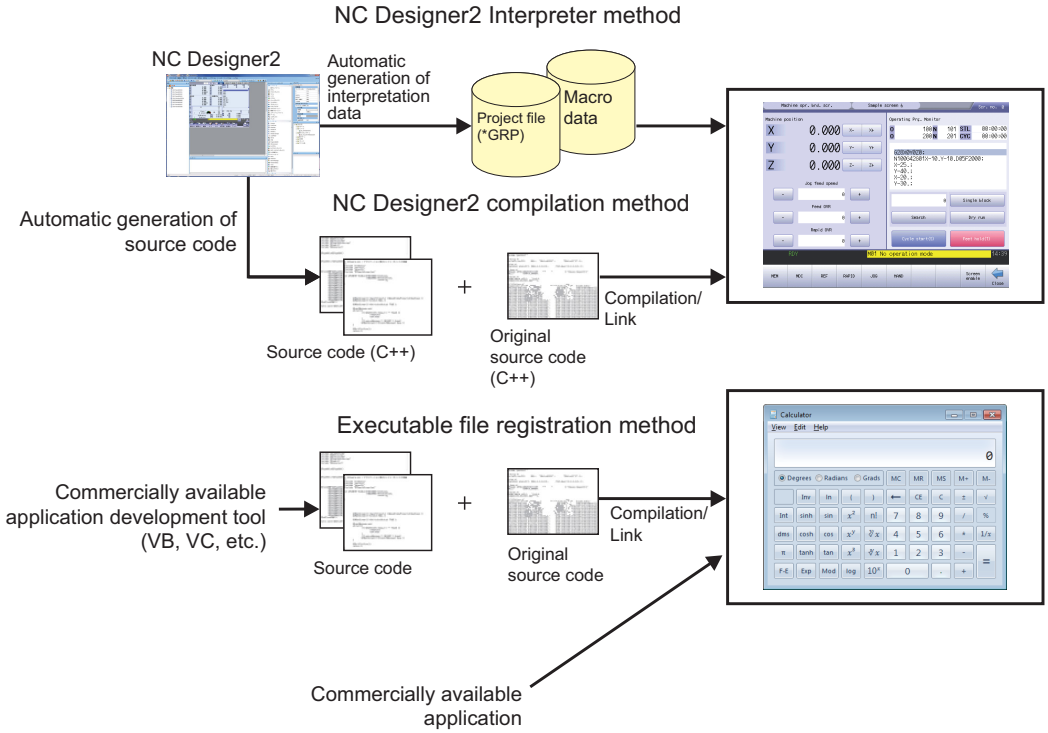
(1) The software sold separately is necessary.

Customization function has four types of assignment, "F0 assignment", "Menu assignment", "Partial-screen customization" and "Selectable-screen customization". Each customized screen can be created and registered with "NC Designer2 interpreter method", "NC Designer2 compilation method" or "Executable file registration method" (*1).

- F0 assignment
  The Customization screen (*2) can be assigned to the function keys (F0, SFP, window display, and window selection). When the function key is pressed, the customized screen assigned is displayed.
- Menu assignment
  The Customization window (*3) can be assigned to a main menu of the monitor screen, setup screen, edit screen, diagnosis screen, or maintenance screen.
  For the monitor screen, setup screen, edit screen, diagnosis screen, or maintenance screen, the display order of main menus can be changed.
  When the registered menu is pressed, the customized window is displayed.
  Also, display/non-display of the customized menu can be changed depending on the conditions.
- Partial-screen customization
  Customized window created with "NC Designer2 interpreter method" or "NC Designer2 compilation method" can be displayed as a part of the standard screen.
- Selectable-screen customization
  Customized window created with "NC Designer2 interpreter method" or "NC Designer2 compilation method" can be displayed in the selectable display area on the monitor screen. Designated customized window (menu assignment) opens by pressing the INPUT key in the same manner as the selectable display of the standard screen.

(*1) "Executable file registration method" cannot be used with M800VS/M80V.
(*2) A user-created HMI screen with NC Designer2 or an executable file prepared by a user.
(Note that the executable file cannot be used with M800VS/M80V.)
(*3) A user-created HMI window with NC Designer2 or an executable file prepared by a user.
(Note that the executable file cannot be used with M800VS/M80V.)

[G-code guidance open, M-code guidance open]
In the G code guidance display area of the edit screen, the guidance of the original G code (G code macro) and M code (M code macro) created by the MTB can be displayed.
By creating a guidance file for G code macro (or M code macro) in HTML format and storing it in a custom folder by the MTB, the corresponding G code (or M code) guidance can be displayed.
If there is both a standard G code guidance file and a G code guidance file created by the MTB, the G code guidance created by the MTB is displayed with priority.


NC Designer2 Interpreter method
NC Designer2
Automatic generation of interpretation data
Project file (*GRP)
Macro data
Automatic generation of source code
NC Designer2 compilation method
Source code (C++)
+
Original source code (C++)
Compilation/ Link
Executable file registration method
Commercially available application development tool (VB, VC, etc.)
Source code
+
Original source code
Compilation/ Link
Commercially available application

17.7.1.1 Customization Data Storage Capacity [MB]

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□ (*1)	□ (*1)	16 (*2)	16 (*2)	□ (*1)	16 (*2)	16 (*2)
L	□ (*1)	□ (*1)	16 (*2)	16 (*2)	□ (*1)	16 (*2)	16 (*2)

(*1) Depending on display unit-side memory space.
(*2) The number indicates the maximum size of the custom screen module (.o).

17.7.1.2 Customization Working Memory Size [MB]

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	6	6	6	6	6	6	6
L	6	6	6	6	6	6	6

17.7.2 User-Defined Key

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

This function allows an arbitrary character string to be assigned to one key and makes it easy to input the fixed phrases.
Register correspondences between SHIFT keys and character strings in a configuration file (keydef.txt).
The maximum number of registerable key types is 8 for ABC layout and 10 for QWERTY layout.
The maximum number of characters used per definition is 256. The definition is disregarded when it is set exceeding the number.

In the configuration file (Keydef.txt), a character string should be defined as follows:

n [character string] ;
n (Registration No.) : 0 to 7 (SHIFT+A to SHIFT+H (ABC layout))
: 0 to 9 (SHIFT+Q to SHIFT+G (QWERTY layout))

(Example)

0[G28X0Y0];	When "SHIFT+A" and a key is input, a character string "G28X0Y0" is input.
1[G28Z0];	When "SHIFT+B" and a key is input, a character string "G28Z0" is input.

17.7.3 EZSocket I/F

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

Note

(1) The software sold separately is necessary.

This middleware makes it easy to develop applications having a Windows interface.
The various functions of the NC unit can be used from a Windows application using VC++ language, VB language and VBA macro language.

17.7.4 APLC Release

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

Note

(1) The software sold separately is necessary.

APLC (Advanced Programmable Logic Controller) release is a function that allows the user-generated C language module to be called from NC.
Control operations that are difficult to express in ladder language can be created with C language.
PLC ladder can easily be customized by converting a part of PLC ladder that has grown complex and bulky into C language module.

There are three methods where APLC is executed. It is possible to use the three in combination.

■ Type 1

By registering the start address of the module to be processed asynchronously with PLC, C language module will be executed in a constant frequency.

■ Type 2

By registering the start address of the module to be processed synchronously with PLC, C language module will be started synchronizing with PLC. Note that C language will not be called if PLC is incorrect, even if the process has been registered.

■ Type 3

By registering the start address of the module to be called from the sequence program and processed, C language module will be started from the sequence program upon S.CALL instruction.

Note

(1) C language module needs to be generated by NC compiler2.

[Hardware configuration]
This function is activated by installing the C language module into the following memory devices.
- SD memory card
- Built-in memory of the control unit

When the module is stored in an SD memory card, the function is activated by inserting the card into the control unit-side SD card interface for M800VW/M80VW Series, while the function is activated by inserting the card into the front-side SD card interface of the display unit for M800VS/M80V Series.
Up to 120 KB data can be stored in built-in memory.
When the C language module is installed into an SD memory card, C language inside the SD memory card is executed. (The module inside the built-in memory is not executed.)

[Software configuration]
The names of directory, file and initialize function, where C language modules are stored, are fixed.

Note

(1) Incorrect hardware or software configuration disables operations.

17.7.5 Custom API Library

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

This function is designed to interface an NC with an application developed by a user.
You can set and refer the data for NC using this function.
Please contact us for details.

17.7.6 MES Interface Library

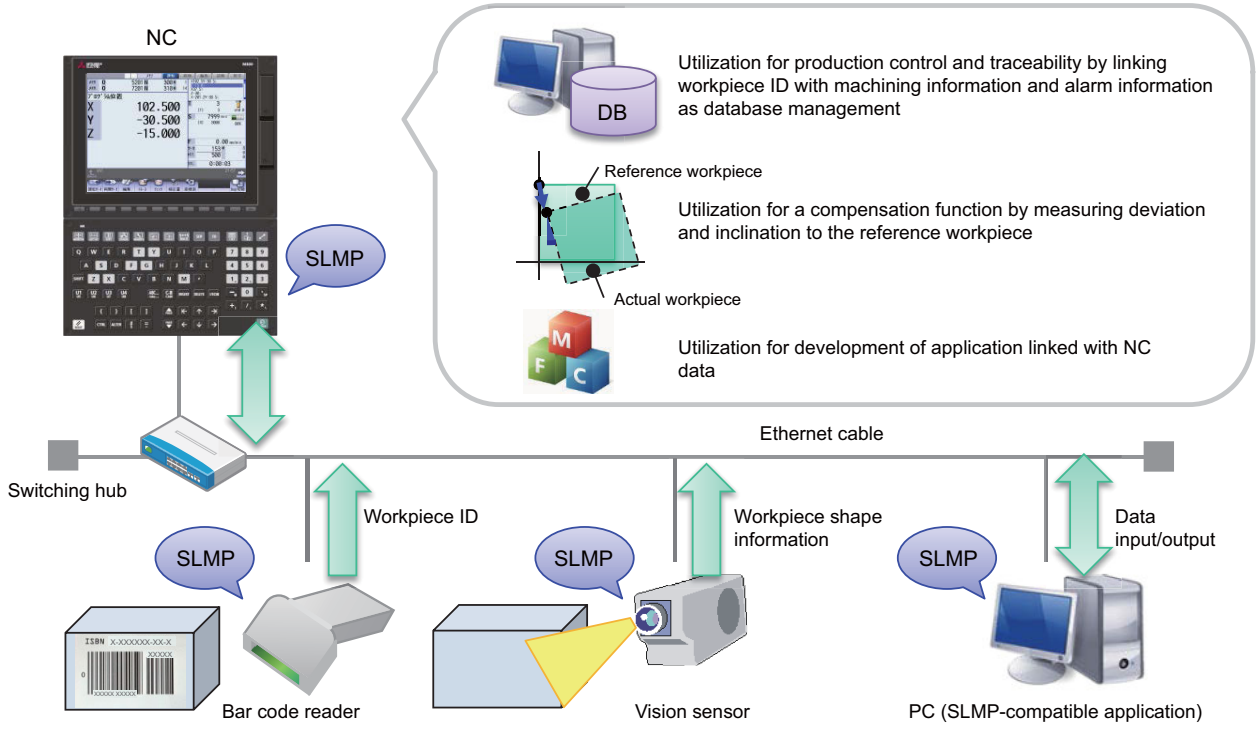
	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

This function enables to link the CNC internal data and the database of information system (manufacturing execution system). Production control and traceability can be ensured with the registered information by registering at machining completion, alarm occurrence, or user's arbitrary timing as needed. The data registered in the database can also be operated with CNC. Conventionally, data was obtained by constantly monitoring CNC from an external computer (information system). However, by using MES interface library function, it is possible to connect without a communication gateway and it allows the communication between CNC and the database at a necessary timing.
This function is supported on CNC and the function range is from obtaining the information at the time of machining completion, alarm occurrence, and user's arbitrary timing to transmitting to the database. Screen application of database and external computer must be prepared by user side.

17.7.7 SLMP Server

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

SLMP server is a function that transfers data using the SLMP between an NC and an external device.
SLMP (Seamless Message Protocol) is a general-purpose protocol to transfer information between the NC and an SLMP-compatible external device through the Ethernet cable and the SLMP is applied to many external devices because it can be easily utilized only by applying S/W.
This function enables easy connection with external devices and read/write NC data through the Ethernet cable. This leads to improve productivity by utilizing external devices and meet broad range of automation needs.



The specifications of SLMP transmitted by an external device are shown below.

Item	Specification
Supported protocols	TCP/IP (UDP is not supported.)
SLMP data code	Binary code (ASCII code is not supported.)
TCP/IP connection waiting port number	30000
Number of TCP/IP connections (Number of external devices connected at the same time)	Up to 4 connections (When multiple connections are made from a single external device, each one is counted as one connection in the total number of connections.)

17.7.8 Numerical Control (CNC) Communication Software FCSB1224W000

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

Note

(1) The software sold separately is necessary.

The Numerical Control (CNC) communication software FCSB1224W000 is designed to help development of an application with Windows interface for Mitsubishi Electric computerized numerical controller.
The product can accelerate development by using OLE interface common to computerized numerical controller without knowing internal processing of the machine.

17.7.11 Motion Control Release

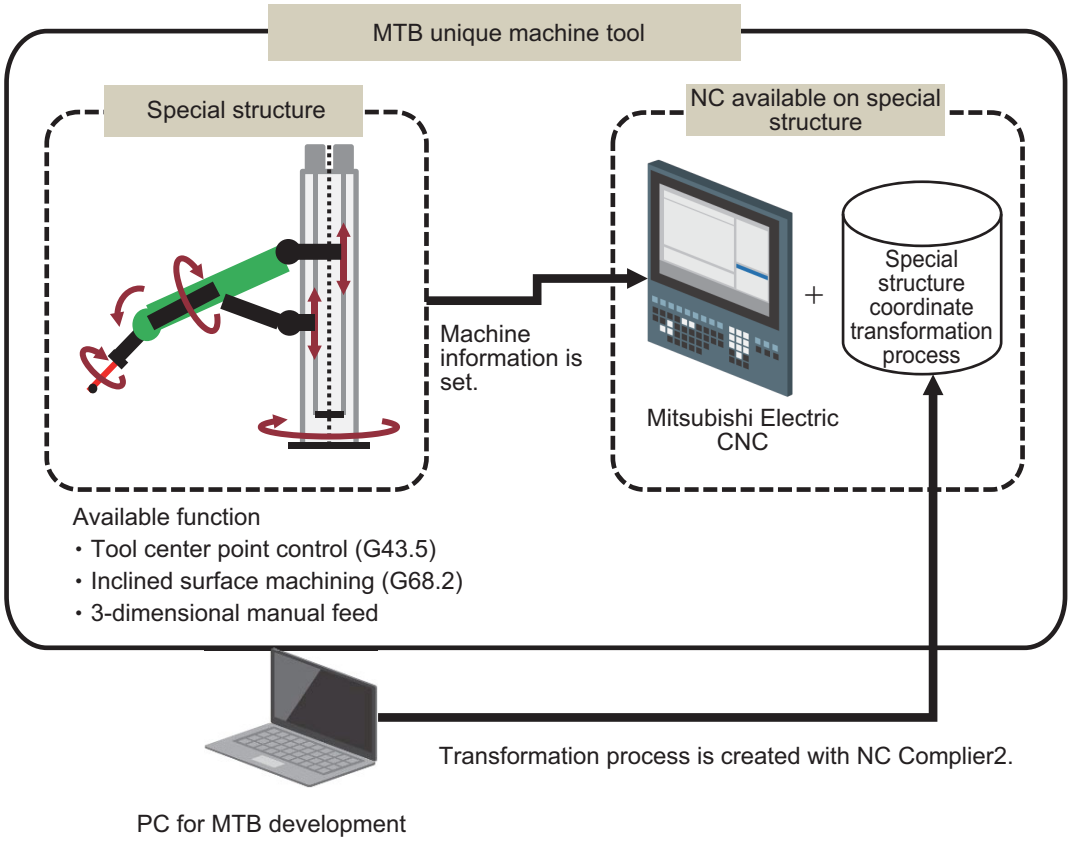
The motion control release enables MTB unique controls by calling functions registered in the MTB-generated C language module from CNC.

17.7.11.1 Coordinate Transformation

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	—	—	—	—	—
L	—	—	—	—	—	—	—

The coordinate transformation realizes the 5-axis control (tool center point control, etc.) in the MTB unique mechanical structure by using coordinate transformation functions created by MTB.
The 5-axis control can be used even on a special structure machine by commanding the 5-axis control from the machining program while add-on modules or function tables for the coordinate transformation are registered.

The following functions are available.
- Tool center point control type 2 (G43.5)
- Inclined surface machining (G68.2)
- 3-dimensional manual feed (table coordinate system/feature coordinate system feed)

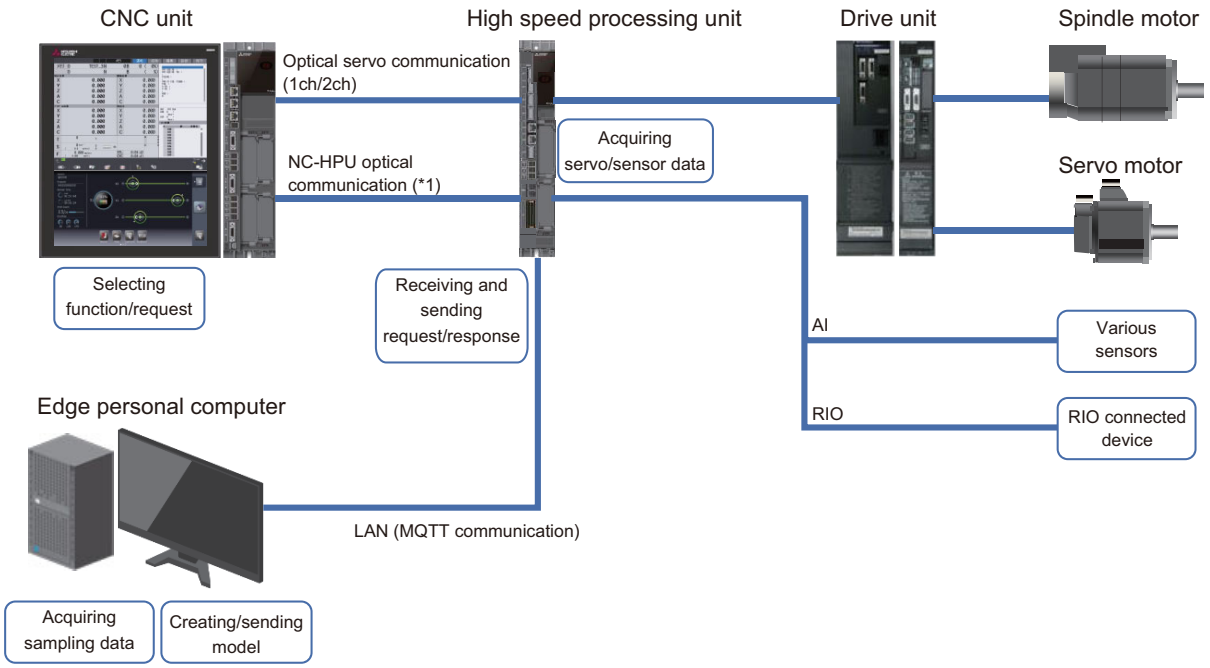


This function is available only on a machine that has a special structure on the tool side. If this function is used on a machine that has a special structure on the table side, the coordinate transformation is not correctly performed. However, CNC cannot judge that the machine does not have the special structure on the table side; therefore, no alarm is output.

17.7.12 Connection with High Speed Processing Unit (HPU)

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	—	—	□	—	—
L	□	□	—	—	□	—	—

The high speed processing unit (HPU) is equipped with functions for sampling data, such as various sensors and I/O signal data in addition to the feedback position and current value of the servo axis, at a high speed.
The sampled data can also be imported into an edge personal computer, and it is possible to add a machine tool builder's original functions, such as the operation status monitoring, machining diagnosis, and preventative maintenance of machine tools.



(*1) NC-HPU optical communication is available only in NC system which has up to 4 part systems and up to 16 axes.

To realize MTB's original functions, the MTB needs to create the following software.

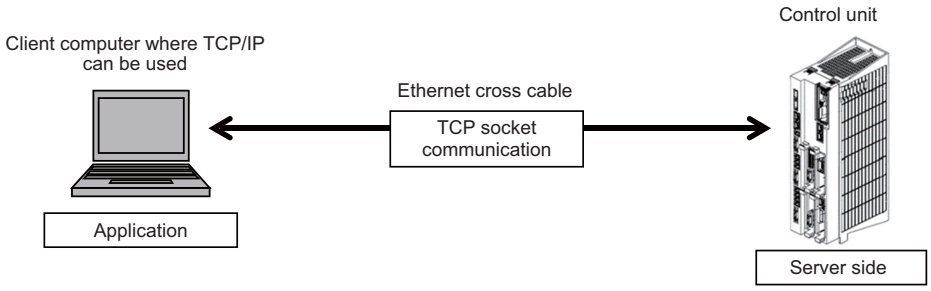
- HPU custom module (Hereinafter, it is called "custom module".)
  This is a software which works in the high speed processing unit. Functions such as data sampling, compensation by model parameter, etc. can be mounted.
- Custom application
  This is a software which works in the edge personal computer. Functions such as creation of model parameter, etc. can be mounted by using data acquired by the custom module.
- Custom screen
  This is created to display an MTB-dedicated screen on a CNC display unit.
- PLC program
  This is created to request sampling and model creation from CNC to a custom module.

17.7.13 Direct Socket Communication I/F

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	△	△	—	—	○	—	—
L	△	△	—	—	○	—	—

This is an I/F function to read/write various CNC data by directly using TCP/IP which is a network communication protocol. The communication is available by connecting an Ethernet cable to M800VW/M80VW control unit.

[Connection example]



To use this function, the optional specification "direct socket communication I/F" is required.
This function can be enabled by parameter setting.

There are the following three communication methods in this function.

General data batch communication method	All the CNC data disclosed on custom API library can be obtained and set at once.
PLC device high-speed communication method	PLC device can be read/written at high speed. The number of device points that can be communicated is limited.
PLC device delay write method	PLC devices can be set all at once. It takes more time to write than with PLC device high-speed communication method, and real-time performance is lower; however, the number of device points is not limited.

- Socket communication is available with up to four client computers.
- Four sockets can be connected to the CNC server.
- For PLC device to be obtained with PLC device high-speed communication method, a delay occurs by at least one scan of sequence program.

Note

(1) With this function, PLC device must be written only through the direct socket communication I/F. Custom API, sequence programs, etc. must not be used to write.

17.8 Others

17.8.1 System Lock

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	Δ	Δ	Δ	Δ	○	○	○
L	Δ	Δ	Δ	Δ	○	○	○

System lock is a function that allows machine tool builders to set the expiration date for use of their machines.
If the cancel code is not entered by the specified deadline, the system forcibly turns OFF the "Servo ready completion" signal to place the machine in an inoperable status.

Note

(1) We do not compensate for any detriment that may be caused when the lock is cancelled illegally.

17.8.2 CNC Remote Operation Tool

17.8.2.1 NC Monitor2

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

Note

(1) The software sold separately is necessary.

NC Monitor2 is the software tool that monitors the condition of NC remotely with a personal computer using a intranet of a plant.
Multiple NCs can be connected and their conditions can be monitored simultaneously.

17.8.2.2 NC Explorer

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	○	○	○	○	○	○	○
L	○	○	○	○	○	○	○

Note

(1) This tool is free of charge. Please contact us.

NC Explorer is a software tool to operate the machining data files of each NC unit connected with a host personal computer by Ethernet connection from the Explorer on the host personal computer.

17.8.3 Automatic Operation Lock

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

Automatic operation lock function prevents the falsification of APLC(C language module: herein after called APLC) by a third party.
Automatic operation will be prohibited when illegal APLC is installed by authenticating APLC with the password for APLC authentication which is registered to NC unit beforehand using this function.
For details on the APLC, refer to " 17.7.4 APLC Release".

17.8.4 Power Consumption Computation

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

Present power consumption and accumulated power consumption amount can be acquired with this function.

[Function to inform power consumption (W)]
The power consumption of servo axis/spindle is informed based on the current information from the drive unit. As for power consumption of other axes, the user needs to set the power consumption values in the NC parameter or PLC signal. The total added with them is also informed.

- Present power consumption

[Function to inform power consumption amount (Wh)]
Accumulated power consumption is informed as power amount. There are the following accumulation functions. Each of them is independent and can be used simultaneously.
Conditions to start/end accumulation of the power consumption, data itself of the accumulated power consumption amount, etc. differ depending on the accumulation function.

- Accumulated power consumption 1 to 4
- Accumulated power consumption history
- Arbitrary section history of accumulated power consumption

17.8.5 EcoMonitorLight Connection

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

Note

(1) EcoMonitorLight sold separately is required.

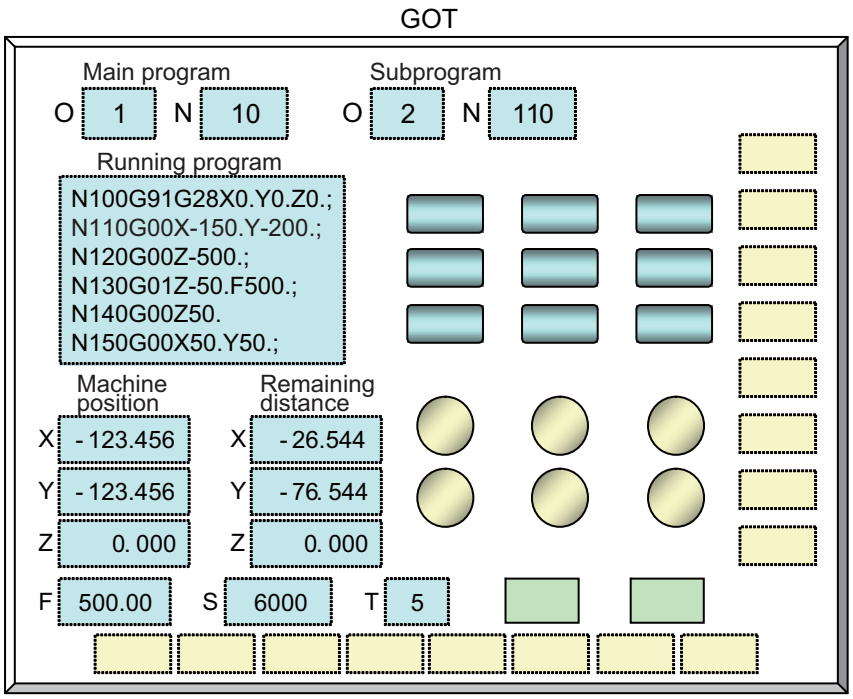
NC system can collect and manage the electric power of the machine tool which is measured by the energy measuring unit "EcoMonitorLight".
The electric power (mainly consumed and regenerated power) of entire machine tool and peripheral devices can be measured by connecting EcoMonitorLight to the main breaker of the machine tool or the breaker of the peripheral devices (motor, coolant, etc.) which configure the machine tool.
By establishing serial communication (Modbus) between the NC unit and EcoMonitorLight using a serial cable, the NC system can collect the power values of the entire machine tool and peripheral devices measured by EcoMonitorLight.
Consequently, the electric power in the machine tool can be finely managed, which enables energy-saving operation.

17.8.6 GOT Window

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	O	O	O	O	O	O	O
L	O	O	O	O	O	O	O

This function is an interface for displaying various CNC data on a GOT that is connected to CNC.
Set a command to CNC from GOT or PLC, and create data that corresponds to the command in the CNC. As the area of up to 20 commands is available, up to 20 commands can be designated at a same time. Also there is an interface that allows CNC to automatically set frequently-used commands at power ON, thereby enabling them to be referred to without a sequence program being created.

[Examples of a screen display]



17.8.8 Laser Processing Control

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	□	□	□	□	□	□	□
L	—	—	—	—	—	—	—

Laser processing control refers generically to the functions that enables the CNC to control the laser oscillator via the laser I/F unit in laser processing of sheet metal.

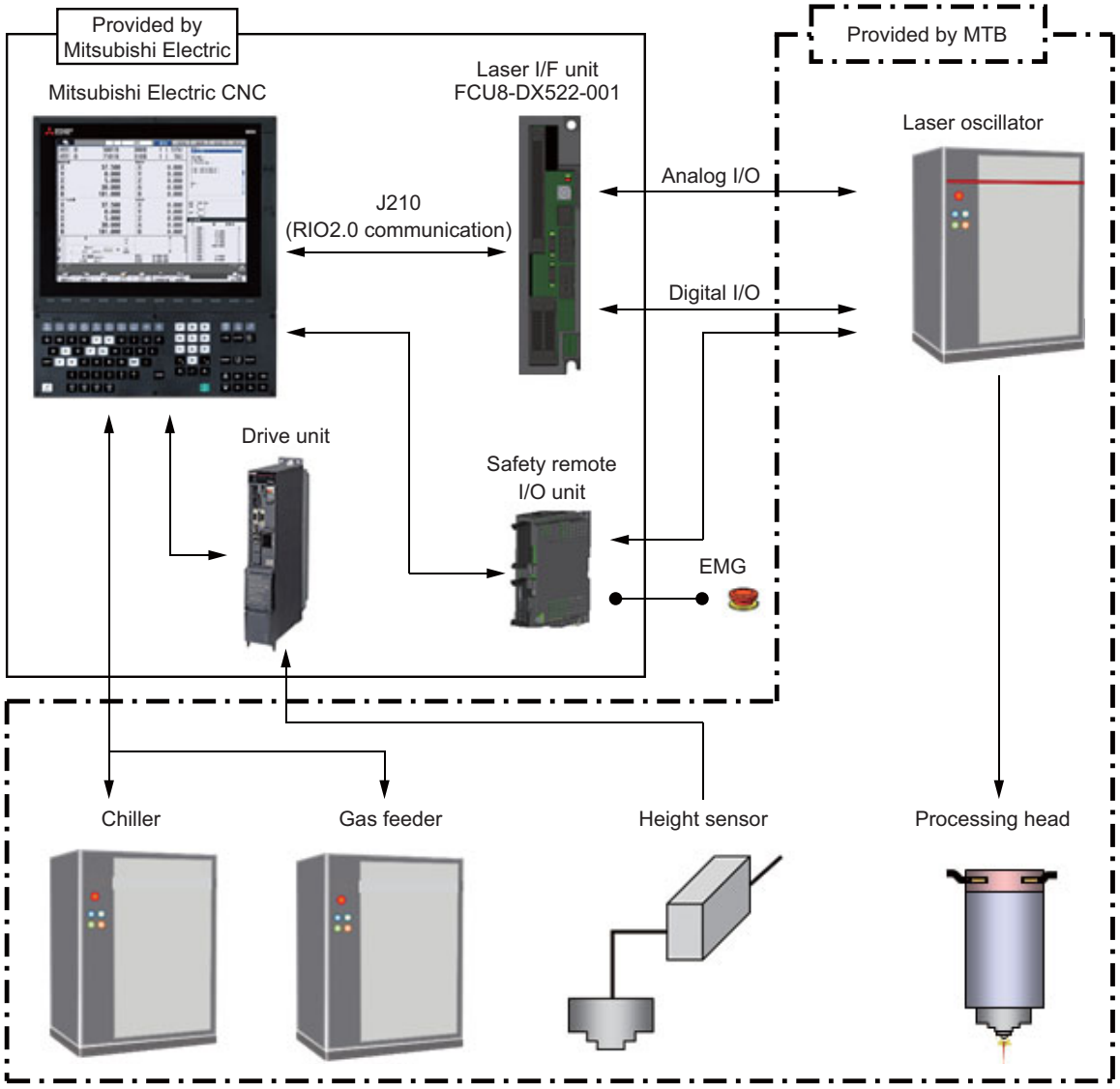
Laser processing function	Details
Laser oscillator output control	The laser oscillator is controlled based on processing conditions, such as laser output (strength), frequency (the number of times of turning ON the laser per second), duty (*1), etc.
Laser oscillator digital I/O control	The digital I/O of the laser oscillator is controlled using PLC devices.
Laser processing condition selection	The processing conditions can be selected using M code from the machining programs by setting multiple combinations of the processing conditions such as laser output, frequency, etc.
DR (dross reduction) control	Laser control appropriate for corners, etc. where speed is varied is performed to reduce dross formation.
F-CUT (fly-cutting) control	Processing of cutting zone and non-cutting zone can be alternately repeated without stopping the axis movement.
Height control	The laser focal length is kept constant even when a deflection occurs in the workpiece, by detecting the distance between the laser head and the machining surface of the workpiece. (A height sensor is required.)
Output calibration control	The laser output can be compensated based on the measurement result of the laser output from the laser head.
Multi-part machining	Multiple same-shape parts can be machined with one G code command. This reduces programming time and improves productivity.

(*1) Ratio of ON time to one-cycle time of turning ON/OFF the laser

The following equipments are required to use the laser processing control.

Required equipment	Remarks
Laser I/F unit FCU8-DX522-001	
Communication cable J210 between CNC and laser I/F unit	
Laser oscillator	Laser oscillator where the control method of the laser I/F unit FCU8-DX522-001 is available
Peripheral devices required for laser processing	Chiller, gas feeder, etc.

[Example of system configuration]



17.8.9 Wireless LAN

	M850VW	M830VW	M850VS	M830VS	M80VW	M80V TypeA	M80V TypeB
M	—	—	○	○	—	○	○
L	—	—	○	○	—	○	○

This function enables the NC to exchange data with devices compliant with wireless LAN standards using wireless communication through the built-in wireless module.
This enables the NC screen to be monitored on a tablet device and allows the peripheral tools that are previously connected by wires to be connected wirelessly, which leads to reduced network wiring cost.

■ 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.

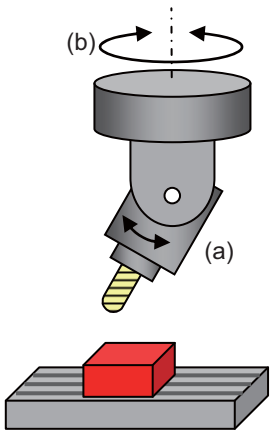
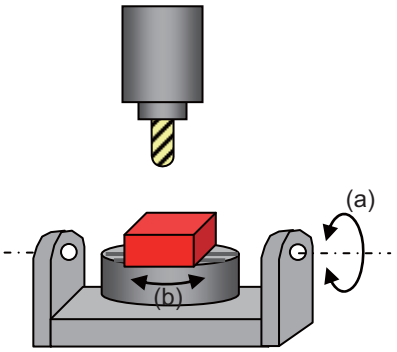
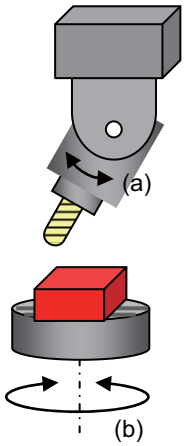
18

Appendix 1: Functions with Advanced Machining

Specific machine configuration is required to use the following functions. Check the machine specifications before use.

- 17.2.11.1 Tool Center Point Control
- 17.2.11.2 Tool Cutting Point Control Command
- 17.2.12 Inclined Surface Machining Command
- 17.2.14 3-Dimensional Tool Radius Compensation (Tool's Vertical-Direction Compensation)
- 17.2.15 Workpiece Installation Error Compensation

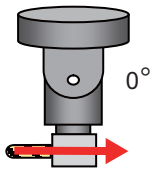
These functions apply to the three types of machine configurations as below:

Type	Tool tilt type	Table tilt type	Combined type
Description	Two rotary axes on tool head side	Two rotary axes on table side	A rotary axis each on tool head side and table side
Example of machines			
Primary rotary axis (a)	The 2nd rotary axis on the tool side	The 1st rotary axis on the table side	Rotary axis on the tool side
Secondary rotary axis (b)	The 1st rotary axis on the tool side	The 2nd rotary axis on the table side	Rotary axis on the table side

Note

(1) These functions cannot be used under the following machine configurations.

- A machine in which three linear axes do not form a right-handed orthogonal coordinate system.
- A machine whose direction from the tool tip to the tool base is not parallel to Z axis (Z axis positive direction) when machine positions of all the rotary axes are "0".

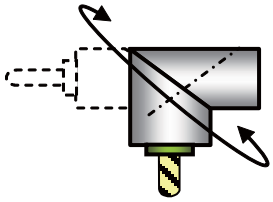


(The arrow indicates the direction of the tool axis.)

- Machine in which the primary rotary axis rotates around the K axis (*1)
- Machine in which the primary rotary axis and the secondary rotary axis rotate around the same axis

(2) The following functions cannot be used with a machine whose rotation center axis of the rotary axis is not parallel to any orthogonal coordinate axis (machine with tilting rotary axis)

- Tool cutting point control
- Workpiece installation error compensation



(3) The following functions can be used with a machine which has one tilting rotary axis.
- Tool center point control
- Inclined surface machining command
- 3-dimensional tool radius compensation (tool's vertical-direction compensation)

(4) The machine configurations that support tilting rotary axes are as follows. Do not use above functions in machine configurations not listed below.
<Tool tilt type>
1st rotary axis: rotates around I axis/2nd rotary axis: rotates around I axis (tilting)
1st rotary axis: rotates around J axis/2nd rotary axis: rotates around I axis (tilting)
1st rotary axis: rotates around K axis/2nd rotary axis: rotates around I axis (tilting)
1st rotary axis: rotates around K axis/2nd rotary axis: rotates around J axis (tilting)
<Combined type>
1st rotary axis: rotates around I axis (tilting)/2nd rotary axis: rotates around J axis
1st rotary axis: rotates around J axis (tilting)/2nd rotary axis: rotates around K axis
<Table tilt type>
1st rotary axis: rotates around I axis (tilting)/2nd rotary axis: rotates around K axis
1st rotary axis: rotates around J axis (tilting)/2nd rotary axis: rotates around K axis

(5) Single axis rotation interpolation method for tool center point control cannot be used in machine configuration with tilting rotary axes.

(6) Tool cutting point control cannot be used in a machine for which the number of simultaneous contouring control axes is 4 or less.

(*1) 3-dimensional tool radius compensation (tool's vertical-direction compensation) cannot be used in a machine in which the secondary rotary axis rotates around the K axis.

Operation when the rotary axis is set with non-ISO polarity

Definition of ISO polarity

ISO polarity	Movement direction and rotation direction of the axis follow the right-handed orthogonal coordinate system (the coordinate system which is defined in ISO standard).
Non-ISO polarity	Movement direction and rotation direction of the axis follow the left-handed orthogonal coordinate system.

The following functions can be used with either ISO polarity or non-ISO polarity.
However, to use the rotary axis installed with non-ISO polarity, all rotary axes must be the same (non-ISO polarity).
- Inclined surface machining command (*1)
- Tool axis direction control
- Tool center point control
- Tool cutting point control
- 3-dimensional manual feed

(*1) "G68.2 P10" command cannot be used.

Note

(1) The following functions cannot be used when a rotary axis is installed with non-ISO polarity.
- 3-dimensional tool radius compensation (tool's vertical-direction compensation)
- Workpiece installation error compensation

Revision History

Revision date	Manual No.	Changes
Mar. 2021	IB(NA)1501610-A	The first edition was created.
Jul. 2021	IB(NA)1501610-B	Contents were revised to correspond to Mitsubishi Electric CNC M800VS/M80V Series. Contents were revised to correspond to Mitsubishi Electric CNC M800V/M80V Series software version A1. The following chapters were added. • 5.1.2 Extended Memory • 6.1.1 Color Display (8.4-Type LCD TFT) • 6.1.2 Color Display (8.4-Type LCD TFT) • 6.1.3 Color Touchscreen Display (15-Type LCD TFT) • 6.2.14 Screenshot Capture • 6.2.20 Remote Desktop Connection • 6.2.21 VNC Server • 6.3.10.18 Vietnamese • 8.1.1.9 Spindle Control with Pulse Train Output • 15.4.4 Servo Tuning Support • 15.4.4.1 NC Analyzer2 • 15.4.10 NC Configurator2 • 17.5.1.3 DI:96/DO:64 • 17.5.1.5 DI:64/DO:48/AO:1 • 17.8.2 CNC Remote Operation Tool • 17.8.2.1 NC Monitor2 • 17.8.2.2 NC Explorer • 17.8.9 Wireless LAN The following chapters were changed. • 1.3.4.2 Display Unit-Side High-Speed Program Server Mode • 5.1 Memory Capacity • 5.2.3 Buffer Correction • 5.2.7 Finish Shape View Programming • 6.3.5 Edit Screen Display • 7.4.3 IP Filter Setting • 8.1.1.7 Spindle-Mode Rotary Axis Control • 12.1.1.1 Subprogram Control [Layers] • 12.1.1.2 Figure Rotation • 12.1.3.1 Fixed Cycle for Drilling • 12.1.3.5 Compound Type Fixed Cycle for Turning Machining • 12.1.8.9 Multi-Part System Program Management • 14.1.8 Rotation Center Error Measurement • 15.1.2 Data Protection Key • 15.2.5 Thermal Detection • 15.2.6 Battery Alarm/Warning • 17.7.1.1 Customization Data Storage Capacity [MB] • 17.7.1.2 Customization Working Memory Size [MB] • 17.7.7 SLMP Server Other mistakes were corrected.

Revision date	Manual No.	Changes
Mar. 2022	IB(NA)1501610-C	Contents were revised to correspond to Mitsubishi Electric CNC M800V/M80V Series software version A3. The following chapters were added. ・6.1.4 Color Touchscreen Display (19-Type LCD TFT) ・7.2.8 Image Input I/F ・13.2.9 Variable Torsion Compensation ・15.3.19 Real-Time 3D Machine Interference Check ・15.4.15 Remote service iQ Care Remote4U ・15.4.15.2 Connection with no RGU ・17.2.25 BiSS Encoder I/F ・17.8.8 Laser Processing Control The following chapters were changed. ・4.7.1 Dwell (Time-Based Designation) ・4.7.2 Dwell (Revolution-Based Designation) ・10.1.13 Workpiece Coordinate System Shift ・13.1.7 Rotation Center Error Compensation ・14.1.3 Manual Tool Length Measurement 1 ・14.1.8 Rotation Center Error Measurement ・17.2.22 External Encoder Position Output I/F The following chapter numbers were changed. ・6.1.8 Separated-Type Color Touchscreen Display (15-Type LCD TFT/Windows10) ・6.1.9 Separated-Type Color Touchscreen Display (19-Type LCD TFT/Windows10) ・6.1.10 Separated-Type Color Touchscreen Display (19-Type Horizontal LCD TFT/Windows10) Other mistakes were corrected.
Mar. 2023	IB(NA)1501610-D	Contents were revised to correspond to Mitsubishi Electric CNC M800V/M80V Series software version A7. The following chapters were added. ・12.1.2.5 Extended Common Variable ・12.1.8.13 Loader Control Part System ・15.4.15.1 Connection with RGU ・17.7.11 Motion Control Release ・17.7.11.1 Coordinate Transformation The following chapters were changed. ・1.1.6 Max. Number of NC Axes in a Part System ・1.1.7 Axis Name Extension ・1.2.2 Max. Number of Part Systems (Main + Sub) ・8.1.1.5 Encoder Input I/F ・12.1.2.2 Machine Tool Builder Macro ・12.1.3.1 Fixed Cycle for Drilling ・12.1.11.1 Rapid Traverse Block Overlap ・12.3.1.2 High-Speed Machining Mode II (G05P2) Maximum [kBPM] ・14.2.1 Tool Life Management ・15.4.13 Security Feature for Windows Display ・17.1.7.1 On-Board Development ・17.2.5 Position Switch ・17.8.4 Power Consumption Computation Other mistakes were corrected.

Revision date	Manual No.	Changes
Jul. 2023	IB(NA)1501610-E	Contents were revised to correspond to Mitsubishi Electric CNC M800V/M80V Series software version A8. The following chapter was added. ・17.2.26 Tool Head Hot Swapping The following chapters were changed. ・4.1.1 Rapid Traverse Rate (m/min) ・4.4.2 Rapid Traverse Constant-Gradient Acceleration/Deceleration ・12.1.2.5 Extended Common Variable ・14.2.1.2 Tool Life Management II ・14.3.7 Direct Robot Control ・15.3.2 Stored Stroke Limit ・15.3.2.1 Stored Stroke Limit I/II ・15.3.2.2 Stored Stroke Limit IB ・15.3.2.3 Stored Stroke Limit IIB ・17.1.2.1 Built-in PLC Basic Function ・17.2.12 Inclined Surface Machining Command ・17.8.8 Laser Processing Control Other mistakes were corrected.
Dec. 2023	IB(NA)1501610-F	Contents were revised to correspond to Mitsubishi Electric CNC M800V/M80V Series software version A9. The following chapters were added. • 12.1.3.9 Text Engraving Cycle • 17.6.11 CC-Link IE TSN (Remote Station) The following chapters were changed. • 4.5.3.2 Pecking Tapping Cycle • 4.5.3.3 Deep-Hole Tapping Cycle • 9.2.1 Tool Radius Compensation • 9.2.3 Tool Nose Radius Compensation (G40/41/42) • 12.1.3.1 Fixed Cycle for Drilling • 12.1.3.6 Compound Type Fixed Cycle for Turning Machining (Type II) • 12.1.7.4 Vibration Cutting Control • 12.3.4 Machining Condition Selection I • 13.2.8 OMR-CC (Optimum Machine Response - Contour Control) • 14.2.1.1 Tool Life Management I • 17.1.7.1 On-Board Development • 17.6.2 PROFIBUS-DP (Master) • 17.6.5 EtherNet/IP Other mistakes were corrected.
Apr. 2024	IB(NA)1501610-G	Contents were revised to correspond to Mitsubishi Electric CNC M800V/M80V Series software version AA. The following chapters were added. • 7.2.9 Measuring Instrument I/F • 15.4.16 Brake Wear Diagnostics • 17.7.12 Connection with High Speed Processing Unit (HPU) The following chapters were changed. • 12.1.2.1 User Macro [Layers] • 12.1.8.5 Control Axis Synchronization Between Part Systems • 12.1.8.9 Multi-Part System Program Management • 12.3.1.2 High-Speed Machining Mode II (G05P2) Maximum [kBPM] • 12.3.3.1 High-Speed High-Accuracy Control I (G05.1Q1) Maximum [kBPM] • 12.3.3.2 High-Speed High-Accuracy Control II (G05P10000) Maximum [kBPM] • 12.3.3.3 High-Speed High-Accuracy Control III (G05P20000) Maximum [kBPM] Other mistakes were corrected.

Revision date	Manual No.	Changes
Oct. 2024	IB(NA)1501610-H	Contents were revised to correspond to Mitsubishi Electric CNC M800V/M80V Series software version B0. The following chapters were added. • 6.3.11 Display Axis Name Selection • 14.3.9 Chatter Suppression • 15.4.2.1 Real-time Data Sampling • 15.4.4.3 On-board Servo Tuning • 15.4.17 NC MachiningAID • 17.1.8.3 Second PLC Constant Extension (300 Points) The following chapters were changed. • 1.1.2 Max. Number of Axes (NC Axes + Spindles + PLC Axes) • 1.1.2.1 Max. Number of NC Axes (In Total for All the Part Systems) • 1.2.2 Max. Number of Part Systems (Main + Sub) • 1.2.2.1 Max. Number of Main Part Systems • 1.2.2.2 Max. Number of Sub Part Systems • 1.3.4.1 Control Unit-Side High-Speed Program Server Mode • 1.3.4.2 Display Unit-Side High-Speed Program Server Mode • 1.3.5 Front-Side SD Card Mode • 1.3.6 Front-Side USB Memory Mode • 4.2.7 G00 Feedrate Designation (,F Command) • 8.1.5.1 Multiple-Spindle Control I • 8.1.10 Tool Spindle Synchronization II (Hobbing) • 12.1.2.2 Machine Tool Builder Macro • 12.1.8.11 Sub Part System Control I • 12.3.1.2 High-speed Machining Mode II (G05P2) Maximum [kBPM] • 12.3.3.1 High-speed High-accuracy Control I (G05.1Q1) Maximum [kBPM] • 14.1.1.1 Skip • 15.3.7 Interference Check III • 17.1.8.1 PLC Constant (150 Points) • 17.2.2 Axis Detachment • 17.7.1 Customization (NC Designer2) • 17.8.4 Power Consumption Computation Other mistakes were corrected.

Revision date	Manual No.	Changes
Apr. 2025	IB(NA)1501610-J	Contents were revised to correspond to Mitsubishi Electric CNC M800V/M80V Series software version B2. The following chapters were added. • 6.2.22 Web Access (NC Screen Remote Browsing) • 12.4.8 Process Analysis • 17.4.7 Extended ZR Device • 17.7.13 Direct Socket Communication I/F • 17.8.6 GOT Window The following chapters were changed. • 1.2.2 Max. Number of Part Systems (Main + Sub) • 1.2.2.2 Max. Number of Sub Part Systems • 2.2.1 Inch/Metric Changeover • 4.5.1 Thread Cutting (Lead/Thread Number Designation) • 4.5.2 Variable Lead Thread Cutting • 5.2.6 Special Program Editing Display for Synchronization Between Part Systems • 6.2.20 Remote Desktop Connection • 7.4.3 IP Filter Setting • 8.1.5.1 Multiple-Spindle Control I • 9.3.3 Number of Tool Offset Sets Allocation to Part Systems • 11.3.4 Program Restart • 12.1.2.4 Variable Commands • 12.1.8.1 Timing Synchronization Between Part Systems • 12.1.8.3.2 Arbitrary Axis Exchange Control • 12.1.8.11 Sub Part System Control I • 12.3.1.2 High-Speed Machining Mode II (G05P2) Maximum [kBPM]] • 12.3.2.2 Multi-Part System Simultaneous High-Accuracy Control • 12.3.3.1 High-Speed High-Accuracy Control I (G05.1Q1) Maximum [kBPM] • 12.3.3.2 High-Speed High-Accuracy Control II (G05P10000) Maximum [kBPM] • 12.3.3.3 High-Speed High-Accuracy Control III (G05P20000) Maximum [kBPM] • 14.1.1.1 Skip • 15.2.6 Battery Alarm/Warning • 15.4.4.3 On-board Servo Tuning • 15.4.17 NC MachiningAID • 17.6.11 CC-Link IE TSN (Remote Station) Other mistakes were corrected.
Aug. 2025	IB(NA)1501610-K	Contents were revised to correspond to Mitsubishi Electric CNC M800V/M80V Series software version B4. The following chapters were changed. • 1.1 Control Axes • 8.1.1.6 Spindle-Mode Servo Motor Control • 8.1.5.2 Multiple-Spindle Control II • 10.1.7 Local Coordinate System • 12.1.3.1 Fixed Cycle for Drilling • 12.1.3.2 Fixed Cycle for Drilling (Type II) • 12.3.4 Machining Condition Selection I • 12.4.8 Process Analysis • 13.1.11 Spatial Error Compensation • 14.2.3 Tool Life Management Set Allocation to Part Systems • 15.4.4.3 On-board Servo Tuning • 17.2.3 Synchronous Control Other mistakes were corrected.

Revision date	Manual No.	Changes
Jan. 2026	IB(NA)1501610-L	Contents were revised to correspond to Mitsubishi Electric CNC M800V/M80V Series software version B8. Chapters 1, 8, 10, 15 and 17 were modified. The main changes are as follows: The following chapter title was changed: • 1.1.4 Max. Number of Indexing PLC Axes The following specifications were changed. • 1.1.2 Max. Number of NC Axes (In Total for All the Part Systems) The number of axes of M80VW for M system The contents were revised as follows. Chapter numbers are indicated in parentheses. • Added the explanation on setting range for spindle position control (spindle/C axis control). (4.1.1, 4.1.2) • Modified the explanation on spindle-mode servo motor control. (8.1.1.6) • Modified the explanation and figure of workpiece coordinate system shift. (10.1.13) • Modified the command format, figure and note of the automatic 1st reference position return. (10.2.2) • Modified the command format and note of the 2nd, 3rd, and 4th reference position return. (10.2.3) • Added the explanation on velocity loop gain adjustment to on-board servo tuning. (15.4.4.3) • Deleted a part of description of MELSEC development tool (GX Developer). (17.1.7.2) Other mistakes were corrected.
May 2026	IB(NA)1501610-M	Contents were revised to correspond to Mitsubishi Electric CNC M800V/M80V Series software version BB. Chapters 1, 3, 12, 17 and 18 were modified. The main changes are as follows. Chapter numbers are indicated in parentheses. The following chapters were added. • 3.2.9 Eccentric Turning • 12.1.7.5 Orbit Machining • 18 Appendix 1: Functions with Advanced Machining The following specifications were changed. • 1.1.2 Max. Number of Axes (NC Axes + Spindles + PLC Axes) The number of axes of M80VW, M80V TypeA and M80V TypeB for M system The number of axes of M80VW and M80V TypeA for L system • 1.1.2.1 Max. Number of NC Axes (In Total for All the Part Systems) The number of axes of M80VW, M80V TypeA and M80V TypeB for M system The number of axes of M80VW and M80V TypeA for L system • 1.1.2.2 Max. Number of Spindles The number of axes of M80V TypeB for M system • 1.1.6 Max. Number of NC Axes in a Part System The number of axes of M80V TypeB for M system The contents were revised as follows • Some contents were deleted and a reference to "18 Appendix 1: Functions with Advanced Machining" was added. (17.2.11.1, 17.2.12) • A reference to "18 Appendix 1: Functions with Advanced Machining" was added. (17.2.11.2, 17.2.14, 17.2.15) 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 2021-2026 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	100-741
Manual No.	IB-1501610

IB(NA)1501610(ENG)-M(2605)MEE
Made in JAPAN

Specifications are subject to change without notice.