

Changes for the Better

MITSUBISHI CNC

Specifications Manual

E70 Series



Introduction

This manual describes the specifications of MITSUBISHI CNC E70 Series.

To safely use this CNC unit, thoroughly study the "Precautions for Safety" on the next page before use.

Details described in this manual

At the beginning of each item, a table indicating it's specification according to the model.

○ : Standard


△ : Optional

□ : Selection

CAUTION

 **The items that are not described in this manual must be interpreted as "not possible".**

 **This manual is written on the assumption that all option functions are added.**




 **Some functions may differ or some functions may not be usable depending on the NC system (software) version.**


General precautions

(1) When the contents of this manual is updated, the version (A, B, ...) on the cover will be incremented.





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, including this manual.











 DANGER When there is a great risk that the user could be subject to fatalities or serious injuries if handling is mistaken.
 WARNING When the user could be subject to fatalities or serious injuries if handling is mistaken.
 CAUTION When the user could be subject to injuries or when physical damage could occur if handling is mistaken.

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

The following signs indicate prohibition and compulsory.

	This sign indicates prohibited behavior (must not do). For example,  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

 **DANGER**




Not applicable in this manual.

 **WARNING**





Not applicable in this manual.

 **CAUTION**

1. 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 option functions are added.
-  Some functions may differ or some functions may not be usable depending on the NC system (software) version.

2. 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 parameter is used to set the temperature rise detection function to invalid, overheating may occur, thereby disabling control and possibly resulting in the axes running out of control, which in turn may result in machine damage and/or bodily injury or destruction of the unit. It is for this reason that the detection function is normally left "valid" for operation. 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 recommends sorting the product and selling the members to appropriate contractors.

- (2) Requirements for "Law for Treatment of Waste and Cleaning"
 - (a) Mitsubishi 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, MELSOFT, GOT, CC-Link, CC-Link/LT and CC-Link IE 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® and Windows® are either trademarks or registered trademarks of Microsoft Corporation in the United States and/or other countries.

CompactFlash and CF are either trademarks or registered trademarks of SanDisk Corporation in the United States and/or other countries.

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.

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(A급) 전자파적합기기로서 판매자 또는 사용자는 이 점을 주의하시기 바라며 가정외의 지역에서 사용하는 것을 목적으로 합니다.

WARRANTY

Please confirm the following product warranty details before using MITSUBISHI 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, we shall provide repair services at no cost through the distributor from which the product was purchased or through a Mitsubishi Electric service provider. Note, however that this shall not apply if the customer was informed prior to purchase of 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 product to the end user, provided the product purchased from us in Japan is installed in Japan (but in no event longer than thirty (30) months, including the distribution time after shipment from Mitsubishi Electric or its distributor).

Note that, for 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; please refer to "2. Service in overseas countries" as will be explained.

[Limitations]

- (1) The customer is requested to conduct an initial failure diagnosis by him/herself, as a general rule. It can also be carried out by us or our service provider upon the customer's request and the actual cost will be charged.
- (2) This warranty applies only when the conditions, method, environment, etc., of use are in compliance with the terms and 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 shall 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 the customer's hardware or software problem
 - (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 may be regarded as avoidable if consumable parts designated in the instruction manual, etc. are duly maintained and replaced
 - (e) any replacement of consumable parts (including a 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 earthquake, lightning, and natural disasters
 - (g) a failure which is unforeseeable under technologies available at the time of shipment of this product from our company
 - (h) any other failures which we are not responsible for or which the customer acknowledges we are not responsible for

2. Service in Overseas Countries

If the customer installs the product purchased from us in his/her machine or equipment, and export it to any country other than where he/she bought it, the customer may sign a paid warranty contract with our local FA center.

This falls under 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 customer purchased the product.

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

Whether during or after the term of warranty, we assume no responsibility for any damages arising from causes for which we are not responsible, any losses of opportunity and/or profit incurred by the customer due to a failure of this product, any damages, secondary damages or compensation for accidents arising under specific circumstances that either foreseen or unforeseen by Mitsubishi Electric, any damages to products other than this product, or compensation for any replacement work, readjustment and startup test run of on-site machines or any other operations conducted by the customer.

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 the use of this product, its applications should be those that may not result in a serious damage even if any failure or malfunction occurs in the product, and a backup or fail-safe function should operate on an external system to the product when any failure or malfunction occurs.
- (2) Mitsubishi 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.

CONTENTS

I General Specifications

1 System Basic Configuration Drawing	I - 1
2 General Connection Diagram	I - 3
3 List of Configuration	I - 7
3.1 List of Units	I - 8
3.2 Durable Parts	I - 9
3.3 Replacements	I - 9
3.4 List of Cables	I - 10
4 Environment Conditions	I - 13
5 Hardware Specifications	I - 15
5.1 Control Unit	I - 16
5.2 Display Unit	I - 26
5.3 Keyboard Unit	I - 27
5.4 Operation Panel I/O Unit.....	I - 29
5.5 Remote I/O Unit	I - 41
5.6 External Power Supply Unit	I - 48
5.7 Manual Pulse Generator	I - 50
5.8 Synchronous Feed Encoder	I - 52
5.9 MITSUBISHI CNC Machine Operation Panel.....	I - 53
5.9.1 MITSUBISHI CNC Machine Operation Panel A.....	I - 53
5.9.2 MITSUBISHI CNC Machine Operation Panel B.....	I - 60
5.10 Exclusive CF Cards for MITSUBISHI CNC.....	I - 64
5.10.1 Precautions for Use of Commercially Available CF Cards.....	I - 64

II Functional Specifications

MITSUBISHI CNC E70 Series Specifications List

(○: Standard, △: Optional, □: Selection)

Class	E70 Series		Page
	M system	L system	
1 Control Axes			II - 1
1.1 Control Axes			II - 2
1.1.1 Number of Basic Control Axes (NC Axes)	○ 3	○ 2	II - 2
1.1.2 Max. Number of Axes (NC Axes + Spindles + PLC Axes)	6	6	II - 2
1.1.2.1 Max. Number of NC Axes (In Total for All the Part Systems)	3	3	II - 2
1.1.2.2 Max. Number of Spindles	1	2	II - 2
1.1.2.3 Max. Number of PLC Axes	2	3	II - 2
1.1.4 Max. Number of PLC Indexing Axes	1	1	II - 2
1.1.5 Number of Simultaneous Contouring Control Axes	3	3	II - 2
1.2 Control Part System			II - 3
1.2.2 Max. Number of Part Systems	○1	○1	II - 3
1.3 Control Axes and Operation Modes			II - 3
1.3.1 Tape (RS-232C Input) Mode	○	○	II - 3
1.3.2 Memory Mode	○	○	II - 3
1.3.3 MDI Mode	○	○	II - 3
1.3.5 Front IC Card Mode	○	○	II - 3
2 Input Command			II - 5
2.1 Data Increment			II - 6
2.1.1 Least Command Increment			II - 6
2.1.1.1 Least Command Increment 1 μm	○	○	II - 6
2.1.1.2 Least Command Increment 0.1 μm	○	○	II - 6
2.1.2 Least Control Increment			II - 7
2.1.2.1 Least Control Increment 0.01μm (10nm)	○	○	II - 7
2.1.2.2 Least Control Increment 0.001μm (1nm)	○	○	II - 7
2.2 Unit System			II - 7
2.2.1 Inch/Metric Changeover	○	○	II - 7
2.2.2 Input Command Increment Tenfold	○	—	II - 8
2.3 Program Format			II - 8
2.3.1 Program Format			II - 8
2.3.1.1 Format 1 for Lathe (G Code List 2, 3)	—	○	II - 8
2.3.1.3 Special Format for Lathe (G Code List 6, 7)	—	○	II - 8
2.3.1.4 Format 1 for Machining Center	○	—	II - 8
2.4 Command Value			II - 9
2.4.1 Decimal Point Input I, II	○	○	II - 9
2.4.2 Absolute/Incremental Command	○	○	II - 10
2.4.3 Diameter/Radius Designation	—	○	II - 12
3 Positioning/Interpolation			II - 13
3.1 Positioning			II - 14
3.1.1 Positioning	○	○	II - 14
3.1.2 Unidirectional Positioning	○	—	II - 15
3.2 Linear/Circular Interpolation			II - 16
3.2.1 Linear Interpolation	○	○	II - 16
3.2.2 Circular Interpolation (Center/Radius Designation)	○	○	II - 17
3.2.3 Helical Interpolation	○	○	II - 19
4 Feed			II - 21
4.1 Feed Rate			II - 22
4.1.1 Rapid Traverse Rate (m/min)	1000	1000	II - 22
4.1.2 Cutting Feed Rate (m/min)	1000	1000	II - 23
4.1.3 Manual Feed Rate (m/min)	1000	1000	II - 24
4.1.4 Rotary Axis Command Speed Tenfold	○	○	II - 24
4.2 Feed Rate Input Methods			II - 25
4.2.1 Feed per Minute	○	○	II - 25
4.2.2 Feed per Revolution	○	○	II - 26
4.2.4 F 1-digit Feed	○	○	II - 27
4.2.5 Manual Speed Command	○	○	II - 28
4.3 Override			II - 28
4.3.1 Rapid Traverse Override	○	○	II - 28
4.3.2 Cutting Feed Override	○	○	II - 28
4.3.3 2nd Cutting Feed Override	○	○	II - 28
4.3.4 Override Cancel	○	○	II - 29
4.4 Acceleration/Deceleration			II - 30
4.4.1 Automatic Acceleration/Deceleration after Interpolation	○	○	II - 30
4.4.2 Rapid Traverse Constant Inclination Acceleration/Deceleration	○	○	II - 32

(○: Standard, △: Optional, □: Selection)

Class	E70 Series		Page
	M system	L system	
4.5 Thread Cutting			II - 36
4.5.1 Thread Cutting (Lead/Thread Number Designation)	○	○	II - 36
4.5.2 Variable Lead Thread Cutting	—	○	II - 38
4.5.3 Synchronous Tapping			II - 39
4.5.3.1 Synchronous Tapping Cycle	○	○	II - 39
4.5.4 Chamfering	—	○	II - 40
4.6 Manual Feed			II - 41
4.6.1 Manual Rapid Traverse	○	○	II - 41
4.6.2 Jog Feed	○	○	II - 42
4.6.3 Incremental Feed	○	○	II - 43
4.6.4 Handle Feed	○	○	II - 43
4.6.5 Manual Feed Rate B	○	○	II - 44
4.7 Dwell			II - 45
4.7.1 Dwell (Time-based Designation)	○	○	II - 45
5 Program Memory/Editing			II - 47
5.1 Memory Capacity			II - 48
5.1.1 Memory Capacity (Number of Programs Stored)			II - 48
5.1.1.5 230kB [600 m] (400 Programs)	○	○	II - 48
5.2 Editing			II - 48
5.2.1 Program Editing	○	○	II - 48
5.2.2 Background Editing	○	○	II - 49
5.2.3 Buffer Correction	○	○	II - 50
6 Operation and Display			II - 51
6.1 Structure of Operation/Display Panel			II - 52
6.1.1 Color Display (8.4-type LCD TFT)	□	□	II - 52
6.2 Operation Methods and Functions			II - 52
6.2.1 Operation Input	○	○	II - 52
6.2.2 Absolute/Incremental Value Setting	○	○	II - 52
6.2.6 Menu List	○	○	II - 53
6.2.7 Display Switch by Operation Mode	○	○	II - 53
6.2.10 Screen Saver	○	○	II - 53
6.2.11 Parameter/Operation Guidance	○(*2)	○(*2)	II - 53
6.2.12 Alarm Guidance	○(*2)	○(*2)	II - 53
6.2.15 Screen Capture	○	○	II - 54
6.2.16 User Selectable Menu Configuration	○	○	II - 54
6.2.20 MTB Selectable Menu Configuration	○	○	II - 54
6.3 Display Methods and Contents			II - 54
6.3.1 Status Display	○	○	II - 54
6.3.2 Clock Display	○	○	II - 54
6.3.3 Monitor Screen Display	○	○	II - 55
6.3.4 Setup Screen Display	○	○	II - 55
6.3.5 Edit Screen Display	○	○	II - 55
6.3.6 Diagnosis Screen Display	○	○	II - 55
6.3.7 Maintenance Screen Display	○	○	II - 55
6.3.8 Additional Language			II - 56
6.3.8.1 Japanese	□	□	II - 56
6.3.8.2 English	○	○	II - 56
6.3.8.3 German	□	□	II - 56
6.3.8.4 Italian	□	□	II - 56
6.3.8.5 French	□	□	II - 56
6.3.8.6 Spanish	□	□	II - 56
6.3.8.7 Chinese			II - 56
6.3.8.7.1 Chinese (Traditional Chinese Characters)	□	□	II - 56
6.3.8.7.2 Chinese (Simplified Chinese Characters)	□	□	II - 56
6.3.8.8 Korean	□	□	II - 57
6.3.8.9 Portuguese	□	□	II - 57
6.3.8.10 Hungarian	□	□	II - 57
6.3.8.11 Dutch	□	□	II - 57
6.3.8.12 Swedish	□	□	II - 57
6.3.8.13 Turkish	□	□	II - 57
6.3.8.14 Polish	□	□	II - 57
6.3.8.15 Russian	□	□	II - 57
6.3.8.16 Czech	□	□	II - 57

(○: Standard, △: Optional, □: Selection)

Class	E70 Series		Page
	M system	L system	
7 Input/Output Functions and Devices			II - 59
7.1 Input/Output Data			II - 60
7.1.1 Machining Program Input / Output	○	○	II - 60
7.1.2 Tool Offset Data Input / Output	○	○	II - 60
7.1.3 Common Variable Input / Output	○	○	II - 60
7.1.4 Parameter Input / Output	○	○	II - 60
7.1.5 History Data Output	○	○	II - 60
7.1.7 System Configuration Data Output	○	○	II - 60
7.2 Input/Output I/F			II - 61
7.2.1 RS-232C I/F	○	○	II - 61
7.2.2 IC Card I/F			II - 61
7.2.2.2 Front IC Card I/F	○	○	II - 61
7.2.3 Ethernet I/F	○	○	II - 61
7.2.6 USB Memory I/F [Up to 2GByte]	○	○	II - 61
8 Spindle, Tool and Miscellaneous Functions			II - 63
8.1 Spindle Functions (S)			II - 64
8.1.1 Spindle Control Functions			II - 64
8.1.1.1 Spindle Digital I/F	○	○	II - 65
8.1.1.2 Spindle Analog I/F	○	○	II - 65
8.1.1.3 Coil Switch	○	○	II - 65
8.1.1.4 Automatic Coil Switch	○	○	II - 65
8.1.1.5 Encoder Input I/F	○	○	II - 66
8.1.2 S Code Output	○	○	II - 67
8.1.3 Constant Surface Speed Control	○	○	II - 68
8.1.4 Spindle Override	○	○	II - 68
8.1.5 Multiple-spindle Control			II - 69
8.1.5.1 Multiple-spindle Control I	—	○	II - 69
8.1.5.2 Multiple-spindle Control II	—	○	II - 70
8.1.6 Spindle Orientation	○	○	II - 70
8.1.7 Spindle Position Control (Spindle/C Axis Control)	○	○	II - 71
8.1.11 Spindle Speed Clamp	○	○	II - 71
8.2 Tool Functions (T)			II - 72
8.2.1 Tool Functions (T Command)	○	○	II - 72
8.3 Miscellaneous Functions (M)			II - 73
8.3.1 Miscellaneous Functions	○	○	II - 73
8.3.2 Multiple M Codes in 1 Block	○	○	II - 73
8.3.3 M Code Independent Output	○	○	II - 74
8.3.4 Miscellaneous Function Finish	○	○	II - 74
8.3.6 Miscellaneous Command High-speed Output	○	○	II - 75
8.4 2nd Miscellaneous Functions (B)			II - 77
8.4.1 2nd Miscellaneous Functions	○	○	II - 77
9 Tool Compensation			II - 79
9.1 Tool Length/Tool Position			II - 80
9.1.1 Tool Length Compensation	○	○	II - 80
9.1.3 Tool Compensation for Additional Axes	—	○	II - 82
9.2 Tool Radius			II - 83
9.2.1 Tool Radius Compensation	○	—	II - 83
9.2.3 Tool Nose Radius Compensation (G40/41/42)	—	○	II - 85
9.2.4 Automatic Decision of Nose Radius Compensation Direction (G46/40)	—	○	II - 86
9.3 Tool Offset Amount			II - 86
9.3.1 Number of Tool Offset Sets			II - 86
9.3.1.3 80 Sets	—	○	II - 86
9.3.1.4 200 Sets	○	—	II - 86
9.3.2 Offset Memory			II - 87
9.3.2.1 Tool Shape/Wear Offset Amount	○	○	II - 87
10 Coordinate System			II - 91
10.1 Coordinate System Type and Setting			II - 92
10.1.1 Machine Coordinate System	○	○	II - 93
10.1.2 Coordinate System Setting	○	○	II - 94
10.1.3 Automatic Coordinate System Setting	○	○	II - 95
10.1.4 Workpiece Coordinate System Selection			II - 96
10.1.4.1 Workpiece Coordinate System Selection (6 Sets)	○	○	II - 96
10.1.4.2 Extended Workpiece Coordinate System Selection (48 Sets) G54.1P1 to P48	○	○	II - 98
10.1.5 External Workpiece Coordinate Offset	○	○	II - 99
10.1.6 Workpiece Coordinate System Preset (G92.1)	—	○	II - 100
10.1.7 Local Coordinate System	○	○	II - 101
10.1.8 Coordinate System for Rotary Axis	○	○	II - 102
10.1.9 Plane Selection	○	○	II - 103
10.1.10 Origin Set/Origin Cancel	○	○	II - 104
10.1.11 Counter Set	○	○	II - 105

(○: Standard, △: Optional, □: Selection)

Class	E70 Series		Page
	M system	L system	
10.2 Return			II - 106
10.2.1 Manual Reference Position Return	○	○	II - 106
10.2.2 Automatic 1st Reference Position Return	○	○	II - 107
10.2.3 2nd, 3rd, 4th Reference Position Return	○	○	II - 109
10.2.4 Reference Position Check	○	○	II - 110
10.2.5 Absolute Position Detection	○	○	II - 111
10.2.6 Tool Exchange Position Return	○	○	II - 112
11 Operation Support Functions			II - 113
11.1 Program Control			II - 114
11.1.1 Optional Block Skip	○	○	II - 114
11.1.3 Single Block	○	○	II - 115
11.2 Program Test			II - 115
11.2.1 Dry Run	○	○	II - 115
11.2.2 Machine Lock	○	○	II - 116
11.2.3 Miscellaneous Function Lock	○	○	II - 116
11.2.4 Graphic Check			II - 117
11.2.4.1 Graphic Check	○	○	II - 117
11.2.5 Graphic Trace			II - 117
11.2.5.1 Graphic Trace	○	○	II - 117
11.2.6 Machining Time Computation	○	○	II - 117
11.3 Program Search/Start/Stop			II - 118
11.3.1 Program Search	○	○	II - 118
11.3.2 Sequence Number Search	○	○	II - 118
11.3.4 Program Restart	○	○	II - 118
11.3.5 Automatic Operation Start	○	○	II - 119
11.3.6 NC Reset	○	○	II - 119
11.3.7 Feed Hold	○	○	II - 120
11.3.8 Search & Start	○	○	II - 120
11.4 Interrupt Operation			II - 120
11.4.1 Manual Interruption	○	○	II - 120
11.4.2 Automatic Operation Handle Interruption	○	○	II - 121
11.4.3 Manual Absolute Switch	○	○	II - 122
11.4.5 Tapping Retract	○	○	II - 123
11.4.6 Manual Numerical Value Command	○	○	II - 124
11.4.8 MDI Interruption	○	○	II - 124
11.4.9 Simultaneous Operation of Manual and Automatic Modes	○	○	II - 125
11.4.10 Simultaneous Operation of JOG and Handle Modes	○	○	II - 125
11.4.11 Reference Position Retract	○	○	II - 125
12 Program Support Functions			II - 127
12.1 Machining Method Support Functions			II - 128
12.1.1 Program			II - 128
12.1.1.1 Subprogram Control	○8layers	○8layers	II - 128
12.1.1.3 Scaling	○	—	II - 130
12.1.2 Macro Program			II - 131
12.1.2.1 User Macro	○4layers	○4layers	II - 131
12.1.2.3 Macro Interruption	○	○	II - 133
12.1.2.4 Variable Command			II - 134
12.1.2.4.2 200 Sets	○	○	II - 134
12.1.3 Fixed Cycle			II - 135
12.1.3.1 Fixed Cycle for Drilling	○	○	II - 136
12.1.3.2 Fixed Cycle for Drilling (Type II)	—	○	II - 141
12.1.3.3 Special Fixed Cycle	○	—	II - 142
12.1.3.4 Fixed Cycle for Turning Machining	—	○	II - 146
12.1.3.5 Compound Type Fixed Cycle for Turning Machining	—	○	II - 151
12.1.3.6 Compound Type Fixed Cycle for Turning Machining (Type II)	—	○	II - 160
12.1.4 Mirror Image			II - 161
12.1.4.3 Mirror Image by G Code	○	—	II - 161
12.1.5 Coordinate System Operation			II - 162
12.1.5.1 Coordinate Rotation by Program	○	—	II - 162
12.1.6 Dimension Input			II - 164
12.1.6.1 Corner Chamfering/Corner R	○	○	II - 164
12.1.6.2 Linear Angle Command	○	○	II - 171
12.1.6.3 Geometric Command	○	○	II - 172
12.1.7 Axis Control			II - 175
12.1.7.3 Circular Cutting	○	—	II - 175
12.1.9 Data Input by Program			II - 176
12.1.9.1 Parameter Input by Program	○	○	II - 176
12.1.9.2 Compensation Data Input by Program	○	○	II - 177
12.1.10 Machining Modal			II - 179
12.1.10.1 Tapping Mode	○	○	II - 179
12.1.10.2 Cutting Mode	○	○	II - 179

(○: Standard, △: Optional, □: Selection)

Class	E70 Series		Page
	M system	L system	
12.2 Machining Accuracy Support Functions			II - 180
12.2.1 Automatic Corner Override	○	○	II - 180
12.2.2 Deceleration Check			II - 181
12.2.2.1 Exact Stop Check Mode	○	○	II - 182
12.2.2.2 Exact Stop Check	○	○	II - 182
12.2.2.3 Error Detection	○	○	II - 182
12.2.2.4 Programmable In-position Check	○	○	II - 183
12.4 Programming Support Functions			II - 184
12.4.1 Playback	○	—	II - 184
12.4.3 Simple Programming	○(*11)	○(*11)	II - 184
12.4.4 G code Guidance	○(*2)	○(*2)	II - 184
13 Machine Accuracy Compensation			II - 185
13.1 Static Accuracy Compensation			II - 186
13.1.1 Backlash Compensation	○	○	II - 186
13.1.2 Memory-type Pitch Error Compensation	○	○	II - 186
13.1.3 Memory-type Relative Position Error Compensation	○	○	II - 187
13.1.4 External Machine Coordinate System Compensation	○	○	II - 187
13.2 Dynamic Accuracy Compensation			II - 188
13.2.1 Smooth High-gain (SHG) Control	○	○	II - 188
13.2.2 Dual Feedback	○	○	II - 190
13.2.3 Lost Motion Compensation	○	○	II - 190
14 Automation Support Functions			II - 191
14.1 Measurement			II - 192
14.1.1 Skip			II - 192
14.1.1.1 Skip	○	○	II - 192
14.1.1.2 Multiple-step Skip	○	○	II - 194
14.1.2 Automatic Tool Length Measurement	○	○	II - 195
14.1.3 Manual Tool Length Measurement 1	○	○	II - 198
14.1.4 Manual Tool Length Measurement 2	○	○	II - 203
14.2 Tool Life Management			II - 205
14.2.1 Tool Life Management			II - 205
14.2.1.1 Tool Life Management I	○	○	II - 205
14.2.1.2 Tool Life Management II	○	○	II - 205
14.2.1.3 Tool Life Management III	○	—	II - 206
14.2.2 Number of Tool Life Management Sets			II - 206
14.2.2.1 80 Sets	—	○	II - 206
14.2.2.2 200 Sets	○	—	II - 206
14.3 Others			II - 207
14.3.1 Programmable Current Limitation	○	○	II - 207
15 Safety and Maintenance			II - 209
15.1 Safety Switches			II - 210
15.1.1 Emergency Stop	○	○	II - 210
15.1.2 Data Protection Key	○	○	II - 210
15.2 Display for Ensuring Safety			II - 210
15.2.1 NC Warning	○	○	II - 210
15.2.2 NC Alarm	○	○	II - 211
15.2.3 Operation Stop Cause	○	○	II - 211
15.2.4 Emergency Stop Cause	○	○	II - 211
15.2.5 Thermal Detection	○	○	II - 212
15.2.6 Battery Alarm/Warning	○	○	II - 212
15.3 Protection			II - 213
15.3.1 Stroke End (Over Travel)	○	○	II - 213
15.3.2 Stored Stroke Limit			II - 213
15.3.2.1 Stored Stroke Limit I/II	○	○	II - 214
15.3.4 Chuck/Tailstock Barrier Check	—	○	II - 216
15.3.5 Interlock	○	○	II - 217
15.3.6 External Deceleration	○	○	II - 217
15.3.9 Door Interlock			II - 218
15.3.9.1 Door Interlock I	○	○	II - 218
15.3.9.2 Door Interlock II	○	○	II - 219
15.3.10 Parameter Lock	○	○	II - 220
15.3.11 Program Protection (Edit Lock B, C)	○	○	II - 220
15.3.12 Program Display Lock	○	○	II - 220
15.3.13 Safety Observation	○	○	II - 221
15.3.14 Vertical Axis Pull-up	○	○	II - 222

(○: Standard, △: Optional, □: Selection)

Class	E70 Series		Page
	M system	L system	
15.4 Maintenance and Troubleshooting			II - 222
15.4.1 Operation History	○	○	II - 222
15.4.2 Data Sampling	○	○	II - 223
15.4.3 NC Data Backup	○	○	II - 223
15.4.5 Servo Tuning Support Tool			II - 223
15.4.5.2 NC Analyzer	○	○	II - 223
15.4.6 Automatic Backup	○	○	II - 223
15.4.7 System Setup	○	○	II - 224
15.4.13 Parameter Setting Tool			II - 225
15.4.13.1 NC Configurator2	○	○	II - 225
16 Drive System			II - 227
16.1 Servo/Spindle			II - 228
16.1.1 Feed Axis			
16.1.1.7 MDS-DJ-V1 (200V)			
16.1.1.7.1 Servo motor: HF**-A48 (260kp/rev)	□	□	
16.1.1.7.2 Servo motor: HF**-A51 (1000kp/rev)	□	□	
16.1.1.7.3 Servo motor: HF-KP**J* (260kp/rev)	□	□	
16.1.1.8 MDS-DM2-SPV2/SPV3 (200V)			
16.1.1.8.1 Servo motor: HF**-A48 (260kp/rev)	□	□	
16.1.1.8.2 Servo motor: HF**-A51 (1000kp/rev)	□	□	
16.1.1.8.3 Servo motor: HF-KP**J* (260kp/rev)	□	□	
16.1.2 Spindle			
16.1.2.9 MDS-DJ-SP(200V)	□	□	
16.1.2.10 MDS-DM2-SPV2/SPV3/SPHV3(200V)	□	□	
16.1.4 Power Supply			
16.1.4.5 AC Reactor for Power Supply	□	□	
16.1.4.6 Ground Plate	□	□	
17 Machine Support Functions			II - 229
17.1 PLC			II - 230
17.1.1 Built-in PLC Processing Mode	○	○	II - 230
17.1.2 PLC Functions			II - 230
17.1.2.1 Built-in PLC Basic Function	○	○	II - 230
17.1.3 PLC Support Functions			II - 230
17.1.3.1 Alarm Message Display	○	○	II - 230
17.1.3.2 Operator Message Display	○	○	II - 231
17.1.3.3 Memory Switch (PLC Switch)			II - 231
17.1.3.3.1 PLC Switch 32 Points	○	○	II - 231
17.1.3.4 Load Meter Display	○	○	II - 231
17.1.3.5 User PLC Version Display	○	○	II - 231
17.1.3.6 Multi-ladder Program Register and Execution	○	○	II - 231
17.1.3.7 Ladder Program Writing during RUN	○	○	II - 232
17.1.3.8 PLC Protection	○	○	II - 232
17.1.4 Built-in PLC Capacity			II - 232
17.1.4.1 Standard PLC Capacity	○8000	○8000	II - 232
17.1.5 Machine Contact Input/Output I/F	○	○	II - 232
17.1.6 Ladder Monitor	○	○	II - 233
17.1.7 PLC Development			II - 233
17.1.7.1 On-board Development	○	○	II - 233
17.1.7.2 MELSEC Development Tool (GX Developer)	○	○	II - 233
17.1.8 PLC Parameter			II - 233
17.1.8.1 PLC Constant (150 Points)	○	○	II - 233
17.2 Machine Construction			II - 234
17.2.1 Servo OFF	○	○	II - 234
17.2.2 Axis Detachment	○	○	II - 235
17.2.4 Inclined Axis Control	—	○	II - 236
17.2.5 Position Switch	○24	○24	II - 237
17.3 PLC Operation			II - 237
17.3.1 Arbitrary Feed in Manual Mode	○	○	II - 237
17.3.3 PLC Axis Control	○	○	II - 238
17.3.5 PLC Axis Indexing	○	○	II - 239
17.4 PLC Interface			II - 240
17.4.1 CNC Control Signal	○	○	II - 240
17.4.2 CNC Status Signal	○	○	II - 241
17.4.3 PLC Window	○	○	II - 243
17.4.4 External Search	○	○	II - 244

(○: Standard, △: Optional, □: Selection)

Class	E70 Series		Page
	M system	L system	
17.5 Machine Contact I/O			II - 244
17.5.1 Additional DI/DO (DI: 32/DO: 32)	□	□	II - 244
17.5.2 Additional DI/DO (DI: 64/DO: 64)	□	□	II - 244
17.5.3 Remote IO 32/32	□	□	II - 244
17.5.4 Remote IO 64/48	□	□	II - 244
17.5.5 MITSUBISHI CNC Machine Operation Panel	△	△	II - 245
17.6 External PLC link			II - 246
17.6.3 CC-Link (Master/Local)	□ (*7)	□ (*7)	II - 246
17.7 Installing S/W for Machine Tools			II - 252
17.7.1 Customization (NC Designer)	○	○	II - 252
17.7.1.1 Customization Data Storage Capacity [MByte]	1	1	II - 253
17.7.1.2 Customization Working Memory Size [MByte]	1	1	II - 253
17.7.1.3 Direct Screen Selection	○	○	II - 253
17.7.3 EZSocket I/F	○	○	II - 253
17.7.4 APLC Release	○	○	II - 253
17.8 Others			II - 254
17.8.1 System Lock	○	○	II - 254

(*2) Guidance data add-on is needed in case of FCU7-MU558-001.

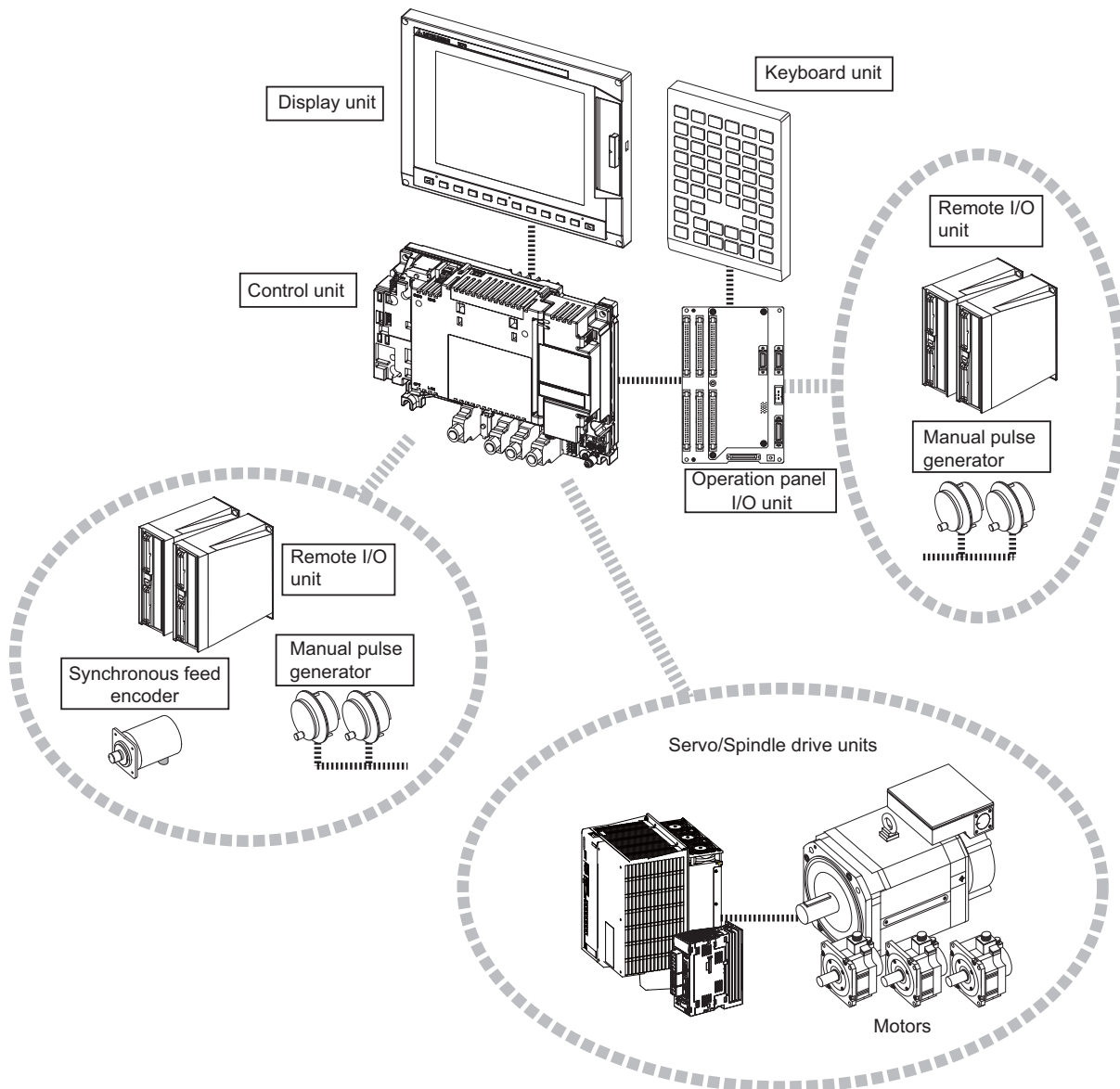
(*7) When HN746 is installed.

(*11) In case of FCU7-MU528-001

I General Specifications



System Basic Configuration Drawing




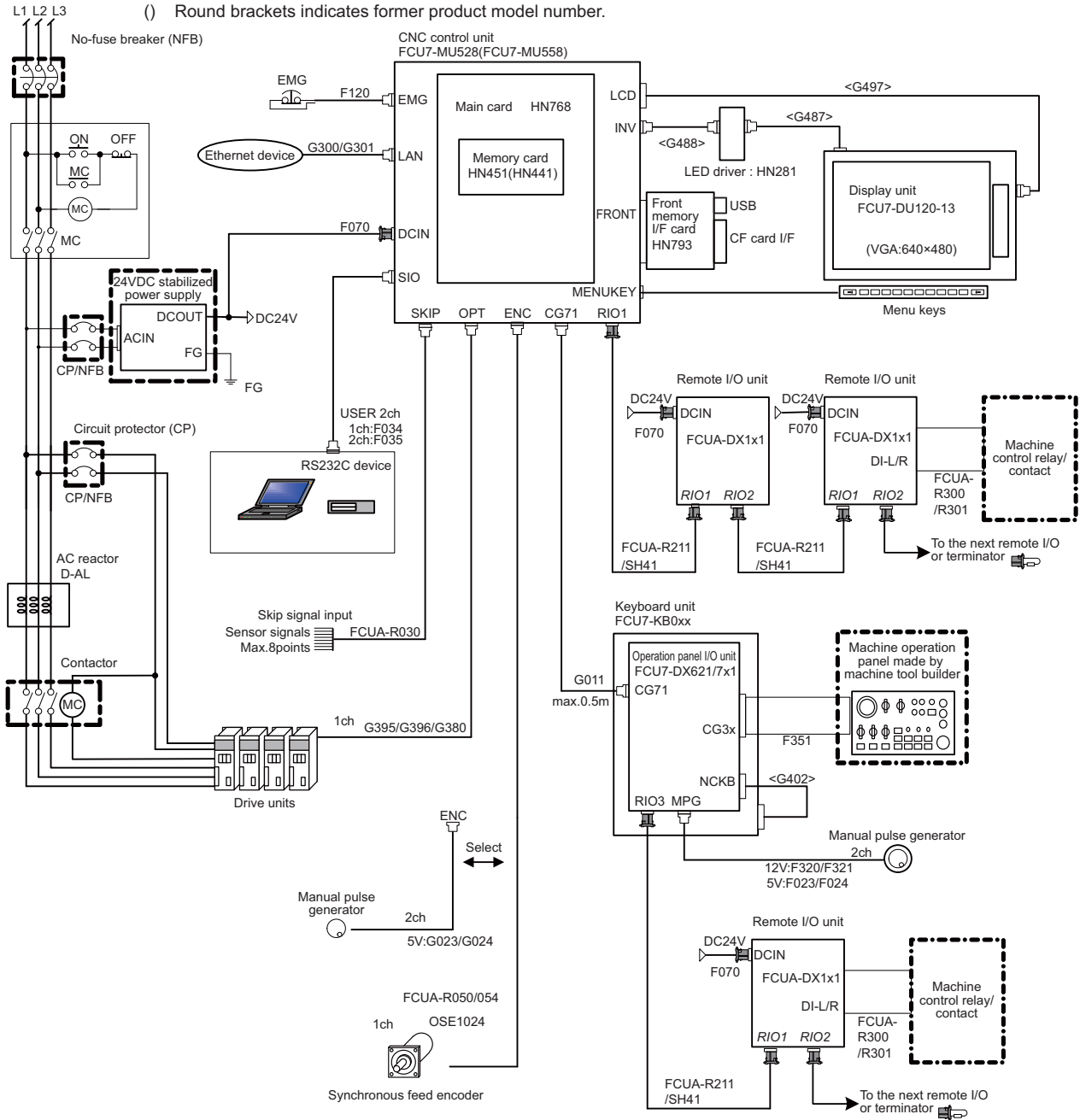
(Note 1) Control unit is mounted on the back side of the display unit.

(Note 2) Operation panel I/O unit is mounted on the back side of the keyboard unit.

(Note 3) For the drive unit configuration, refer to the Instruction Manual of the drive unit you use.



General Connection Diagram

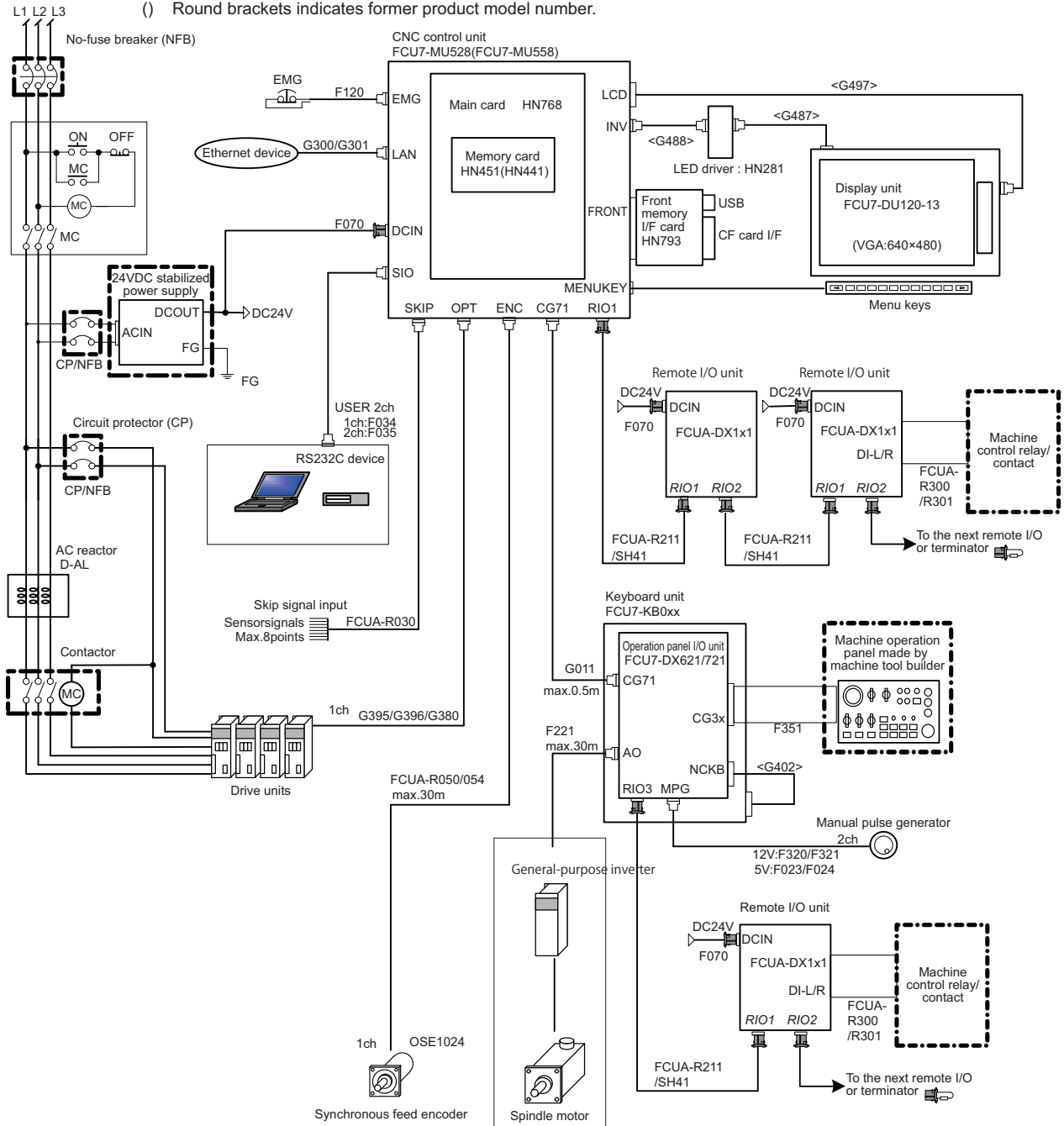
-  Dotted lines indicate the sections prepared by the machine tool builder.
- <> Angle brackets indicates attached cable of unit.
- () Round brackets indicates former product model number.



(Note 1) For information on how to connect the drive unit, refer to the drive unit's manual.

[Analog Spindle Configuration]

 Dotted lines indicate the sections prepared by the machine tool builder.
 Angle brackets indicates attached cable of unit.
 () Round brackets indicates former product model number.



(Note 1) For information on how to connect the drive unit, refer to the drive unit's manual.

List of Configuration

3.1 List of Units

Classification	Type	Components	Remarks
[Control unit]			
NC functions and display controller	FCU7-MU528	Main control card Memory card (HN451) Front side memory I/F card	Export Tarde Control Order and Foreign Exchange Order noncompliant unit E70 S/W version L0 or later
NC functions and display controller	FCU7-MU558	Main control card Memory card (HN441) Front side memory I/F card	Export Tarde Control Order and Foreign Exchange Order noncompliant unit
[Display unit]			
8.4-type color TFT (VGA:640*480)	FCU7-DU120-13	LCD panel Backlight driver Menu keys Driver cable LED backlight cable LCD cable	Front side memory I/F is normally equipped with the control unit
[Keyboard unit]			
Keyboard for 8.4-type display unit Sheet keys	FCU7-KB024	Escutcheon, key switch G402 cable	ONG layout (for M system/L system, XYZ)
Keyboard for 8.4-type display unit Lathe system sheet keys	FCU7-KB025	Escutcheon, key switch G402 cable	ONG layout (for L system, XZF)
[Operation panel I/O unit]			
DI 24V/0V common input DO Source output AO Analog output	FCU7-DX621	Base card Terminator (R-TM) Add-on card	DI: 64-points 24V/0V common type DO: 48-points source type MPG:2ch AO: 1 point Occupied stations (fixed): 1, 2, 3, 7, 8 RIO3 extensible stations: 4, 5, 6
DI 24V/0V common input DO Source output	FCU7-DX711	Base card Terminator (R-TM)	DI: 64-points 24V/0V common type DO: 64-points source type MPG:2ch Occupied stations (fixed): 1, 2, 7, 8 RIO3 extensible stations: 3, 4, 5, 6
DI 24V/0V common input DO Source output AO Analog output	FCU7-DX721	Base card Terminator (R-TM) Add-on card	DI: 96-points 24V/0V common type DO: 80-points source type MPG:2ch AO: 1 point Occupied stations (fixed): 1, 2, 3, 7, 8 RIO3 extensible stations: 4, 5, 6
DI 24V/0V common input DO Source output	FCU7-DX731	Base card Terminator (R-TM) Add-on card	DI: 96-points 24V/0V common type DO: 96-points source type MPG:2ch Occupied stations (fixed): 1, 2, 3, 7, 8 RIO3 extensible stations: 4, 5, 6
[Remote I/O unit]			
DI 24V/0V common input DO Source output	FCUA-DX101	RX312	DI: 32-points 24V/0V common type (photo coupler insulation) DO: 32-points source type (non-insulation) Number of occupied stations: 1
DI 24V/0V common input DO Source output	FCUA-DX111	RX312+RX322-1	DI: 64-points 24V/0V common type (photo coupler insulation) DO: 48-points source type (non-insulation) Number of occupied stations: 2
DI 24V/0V common input DO Source output AO Analog output	FCUA-DX121	RX312+RX322	DI: 64-points 24V/0V common type (photo coupler insulation) DO: 48-points source type (non-insulation) AO: 1 point Number of occupied stations: 2
DI 24V/0V common input DO Source output AI Analog input AO Analog output	FCUA-DX141	RX312+RX341	DI: 32-points 24V/0V common type (photo coupler insulation) DO: 32-points source type (non-insulation) AI: 4 points AO: 1 point Number of occupied stations: 2

Classification	Type	Components	Remarks
[External power supply unit]			
External power supply with power supply ON/OFF function	PD25	Power supply card Case set	Input 200VAC Output 24VDC (3A)
[Manual pulse generator]			
5V Manual pulse generator	UFO-01-2Z9	UFO-01-2Z9 (Produced by NIDEC NEMICON)	Input 5VDC 100pulse/rev
12V Manual pulse generator	HD60C	HD60C	Input 12VDC 25pulse/rev
[Encoder]			
Synchronous feed encoder	OSE1024-3-15-68	OSE1024-3-15-68	Input 5VDC 1024pulse/rev
[CC-Link unit]			
CC-Link	FCU7-HN746	HN746	CC-Link × 1ch
[MITSUBISHI CNC machine operation panel]			
MITSUBISHI CNC machine operation panel A	FCU7-KB921	Escutcheon, key switch control card	Mitsubishi standard 55 key
MITSUBISHI CNC machine operation panel B	FCU7-KB926	Escutcheon, Switch	
[Cable connector sets]			
For ENC,SKIP,SIO,MPG,AO	FCUA-CS000	Connector (10120-3000VE,2pcs), Shell kit (10320-52F0-008,2pcs)	
For RIO	FCUA-CS301	Connector (7940-6500SC,4pcs.), Strain relief (3448-7940,2pcs.)	
For PD25/PD27 ACIN	FCUA-CN200	Connector (2-178128-3), Contact (1-175218-5,3pcs.)	
For PD25/PD27 DCOUT	3-178127-6 1-175218-5	Connector (3-178127-6), Contact (1-175218-5,6pcs.)	
For PD25/PD27 ON/OFF	1-178288-5 1-175218-5	Connector (1-178288-5), Contact (1-175218-5,6pcs.)	
For EMG F120	005057-9403 0016020103	Connector (005057-9403), Contact (0016020103,3pcs.)	
For RIO	FCUA-CN211	Connector (1-178288-3), Contact (1-175218-2,3pcs.)	
For DCIN,F070	FCUA-CN220	Connector (2-178288-3), Contact (1-175218-5,3pcs)	
For DIO	FCUA-CN300	Connector (7940-6500SC,2pcs.)	
[Original manufactured memory card]			
Exclusive CF cards for MITSUBISHI CNC 256MB	FCU7-CF256M	FCU7-CF256M	256MB capacity
Exclusive CF cards for MITSUBISHI CNC 2GB	FCU7-CF002G	FCU7-CF002G	2GB capacity

(Note 1) Operation panel I/O unit can be mounted on the back side of the keyboard unit.

(Note 2) Operation panel I/O units for 700 Series (FCU7-DX67x/ FCU7-DX77x) are not available.

(Note 3) DI: Digital input signals, DO: Digital output signals, AI: Analog input signals, AO: Analog output signals

3.2 Durable Parts

Durable parts	Part type
Control unit battery	Q6BAT
Backlight for FCU7-DU120-13(*)	84LHS16
Key sheet for FCU7-KB024	N330B532G51
Key sheet for FCU7-KB025	N330B532G52

(*) Contact the Service Center, Service Station, Sales Office or delayer for repairs or part replacement.

3.3 Replacements

Replacements	Part type
Protection fuse	LM40

3.4 List of Cables

Type	Application	Length (m) of cables provided by Mitsubishi	Max. cable length
CNP2E-1-xM	Motor side PLG cable	2, 3, 4, 5, 7, 10, 15, 20, 25, 30	30m
CNP3EZ-2P-xM	Spindle side detector cable	2, 3, 4, 5, 7, 10, 15, 20, 25, 30	30m
CNP3EZ-3P-xM	Spindle side detector cable	2, 3, 4, 5, 7, 10, 15, 20, 25, 30	30m
CNV22J-K1P-0.3M	Detector extension cable for HF-KP motor	0.3	0.3m
CNV22J-K2P-0.3M	Detector extension cable for HF-KP motor	0.3	0.3m
CNV2E-8P-xM	Motor side detector cable (for A74(N)/A51/A48)/ Ball screw side detector cable	2, 3, 4, 5, 7, 10, 15, 20, 25, 30	30m
CNV2E-9P-xM	Motor side detector cable (for A74(N)/A51/A48)/ Ball screw side detector cable	2, 3, 4, 5, 7, 10, 15, 20, 25, 30	30m
CNV2E-HP-xM	MDS-B-HR unit cable	2, 3, 4, 5, 7, 10, 15, 20, 25, 30	30m
CNV2E-K1P-xM	Detector cable for HF-KP motor (load side angle)	2, 3, 5, 7, 10	10m
CNV2E-K2P-xM	Detector cable for HF-KP motor (reverse load side angle)	2, 3, 5, 7, 10	10m
DG21-xM	Battery cable (For drive unit - battery unit)	0.3, 0.5, 1, 5	5m
DG22-xM	Battery cable (For servo drive unit - servo drive unit) * This cable is required to supply the power from the battery unit to multiple drive units.	0.3, 0.5, 1, 2, 3, 5, 7, 10	10m
DG23-xM	Battery cable (For servo drive unit -battery box)	0.3, 0.5, 1, 2, 3, 5, 7, 10	10m
DG24-xM	5V spply/DO output cable (For servo drive unit -battery box)	0.3, 0.5, 1, 2, 3, 5, 7, 10	10m
DG25-xM	Battery cable (For servo drive unit -battery box)	0.3, 0.5, 1, 2, 3, 5, 7, 10	10m
F023 LxM	Manual pulse generator cable (5V): 1ch (for connection to operation panel I/O unit)	1, 2, 3, 5, 8, 10, 15, 20	20m
F024 LxM	Manual pulse generator cable (5V): 2ch (for connection to operation panel I/O unit)	1, 2, 3, 5, 8, 10, 15, 20	20m
F034 LxM	RS-232C I/F cable: 1ch	0.5, 1, 2, 3, 5, 8, 10	15m (*)
F035 LxM	RS-232C I/F cable: 2ch	0.5, 1, 2, 3, 5, 8, 10	15m (*)
F070 LxM	24VDC power cable	0.5, 1.5, 3, 5, 8, 10, 15, 20	30m
F110 LxM	24VDC power cable for PD25	0.5, 1.5, 3, 5, 8, 10, 15	15m
F120 LxM	Emergency stop cable	0.5, 1.5, 3, 5, 8, 10, 15, 20	30m
F170 LxM	ON/OFF switch cable for PD25	0.5, 1.5, 3, 5, 8, 10, 15	15m
F221 LxM	Analog output cable	1, 2, 3, 5, 8, 10, 15, 20	30m
F320 LxM	Manual pulse generator cable (12V): 1ch (for connection to operation panel I/O unit)	1, 2, 3, 5, 8, 10, 15, 20	50m
F321 LxM	Manual pulse generator cable (12V): 2ch (for connection to operation panel I/O unit)	1, 2, 3, 5, 8, 10, 15, 20	50m
F351	DI/DO cable (one side connector) (for operation panel I/O unit)	3	50m
FCUA-R030-xM	SKIP input	3, 7	20m
FCUA-R031-xM	Analog input/output	2, 3, 7	30m
FCUA-R050-xM	Encoder input (straight, with connector)	5	30m
FCUA-R054-xM	Encoder input (right angle, with connector)	3, 5, 10, 15, 20	30m
FCUA-R211-xM	Remote I/O (with terminal block)	0.3, 1, 2, 3, 5, 8, 10, 15, 20	30m (*)
FCUA-R300	DI/DO cable (one side connector) (for remote I/O unit)	3	50m
FCUA-R301-xM	DI/DO cable (both side connectors) (for remote I/O unit)	1, 2, 3, 5	50m
G011 LxM	Operation panel I/O interface cable	0.5	0.5m
G023 LxM	Manual pulse generator cable (5V): 1ch (for connection to control unit)	1, 2, 3, 5, 8, 10, 15, 20	20m (*)
G024 LxM	Manual pulse generator cable (5V): 2ch (for connection to control unit)	1, 2, 3, 5, 8, 10, 15, 20	20m (*)
G071 xM	DC24V relay cable for MITSUBISHI CNC machine operation panel	0.5	0.5m
G300 LxM	LAN cross cable (Shielded cable is recommended when the length will be 1m or more)	1, 3, 5, 10	10m
G301 LxM	LAN straight cable (Shielded cable is recommended when the length will be 1m or more)	1	1m
G380 LxM	Optical communication cable (PCF type with reinforced sheath) (for wiring outside of the panel)	5, 10, 12, 15, 20, 25, 30	30m
G395 LxM	Optical communication cable (POF type with reinforced sheath) (for wiring outside of the panel)	3, 5, 7, 10	10m
G396 LxM	Optical communication cable (POF type without reinforced sheath) (for wiring inside of the panel)	0.3, 0.5, 1, 2, 3, 5	10m
G460 xM	Cable between MITSUBISHI CNC machine operation panel A and MITSUBISHI CNC machine operation panel B	0.5	0.5m

Type	Application	Length (m) of cables provided by Mitsubishi	Max. cable length
MR-BKS1CBLxM-A1-H	Brake cable for HF-KP motor (load side angle)	2, 3, 5, 7, 10	10m
MR-BKS1CBLxM-A2-H	Brake cable for HF-KP motor (reverse load side angle)	2, 3, 5, 7, 10	10m
MR-PWS1CBLxM-A1-H	Power cable for HF-KP motor (load side angle)	2, 3, 5, 7, 10	10m
MR-PWS1CBLxM-A2-H	Power cable for HF-KP motor (reverse load side angle)	2, 3, 5, 7, 10	10m
R-TM	Remote I/O Interface terminator	-	-
SH21 xM	Power supply communication cable	0.35, 0.5, 1, 2, 3, 5, 10, 15, 20, 30	30m
SH41 xM	Remote I/O (between units in a panel)	0.3, 0.5, 0.7	1m (*)

(Note 1) Asterisks "x" in type columns indicate cable length (unit: m).

(Note 2) Lengths indicated with an asterisk (*) in the max. cable length column indicate the maximum cable length when connecting via other unit.



4

Environment Conditions

I General Specifications

Item	Unit name		Control unit	Display unit	Keyboard unit	Operation panel I/O unit	Machine operation panel
	Type		FCU7-MU528 (FCU7-MU558)	FCU7-DU120-13	FCU7-KB024/025	FCU7-DX621/711/ 721/731	FCU7-KB921/926
General specifications	Ambient temperature	During operation	0 to 55C°				
		During storage	-20 to 60C°				
	Ambient humidity	Long term	10 to 75% RH (with no dew condensation)				
		Short term	10 to 95% RH (with no dew condensation) (Note 1)				
	Vibration resistance		4.9m/s ² or less (during operation)				
	Shock resistance		29.4m/s ² or less (during operation)				
	Working atmosphere		No corrosive gases, dust or oil mist				
Required power specifications	Power voltage		24VDC ±5% Ripple noise 200mV (P-P)	3.3V/12VDC	5VDC	3.3V/5VDC	24VDC ±5%(Note 7)
	Power capacity		24V 2.5A	-	-	- (Note 2)	0.25A(Note 7)
	Instantaneous stop tolerance time		20ms			-	20ms(min)(Note 7)
Others	Heating value	(max.)	12.0W	3.3W	1.0W	Control section: 5.0W (Note 3)	6W
	Mass	(kg)	1.0	1.5	0.8	0.4	FCU7-KB921:1.2 FCU7-KB926:0.5
	Outline dimension	(mm)	235(width) x 173(height) x 73(depth) (Depth from the plate mounting surface: 60)	260(width) x 200(height)	140(width) x 200(height)	120(width) x 180(height)	FCU7-KB921: 260(W) x 140(H) FCU7-KB926: 140(W) x 140(H)

Item	Unit name		Remote I/O unit			
	Type		FCUA-DX101	FCUA-DX111	FCUA-DX121	FCUA-DX141
General specifications	Ambient temperature	During operation	0 to 55C°			
		During storage	-20 to 60C°			
	Ambient humidity	Long term	10 to 75% RH (with no dew condensation)			
		Short term	10 to 95% RH (with no dew condensation) (Note 1)			
	Vibration resistance		4.9m/s ² or less (during operation)			
	Shock resistance		29.4m/s ² or less (during operation)			
Working atmosphere		No corrosive gases or dust				
Required power specifications	Input power voltage		24VDC±5% Ripple noise 200mV (P-P)			
	Power capacity		24V 0.7A (Note 4)	24V 1.5A (Note 4)	24V 0.7A (Note 4)	
	Instantaneous stop tolerance time		-			
Others	Heating value	(max.)	25W (Note 5)	30W (Note 5)	30W (Note 5)	
	Mass	(kg)	0.5	0.6	0.6	0.6

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

(Note 2) For the current value of the I/O circuit, calculate with the number of points used and load.

(Note 3) For the heating value of the I/O circuit, calculate with the number of points used.

(Note 4) Allows only the amount to be consumed by control circuit.

(Note 5) Differs according to the number of machine input operation points and the load and number of points connected to the machine output. The maximum value applies when all points are ON.

(Note 6) These units, which are open equipments, must be installed within a sealed metal control panel.

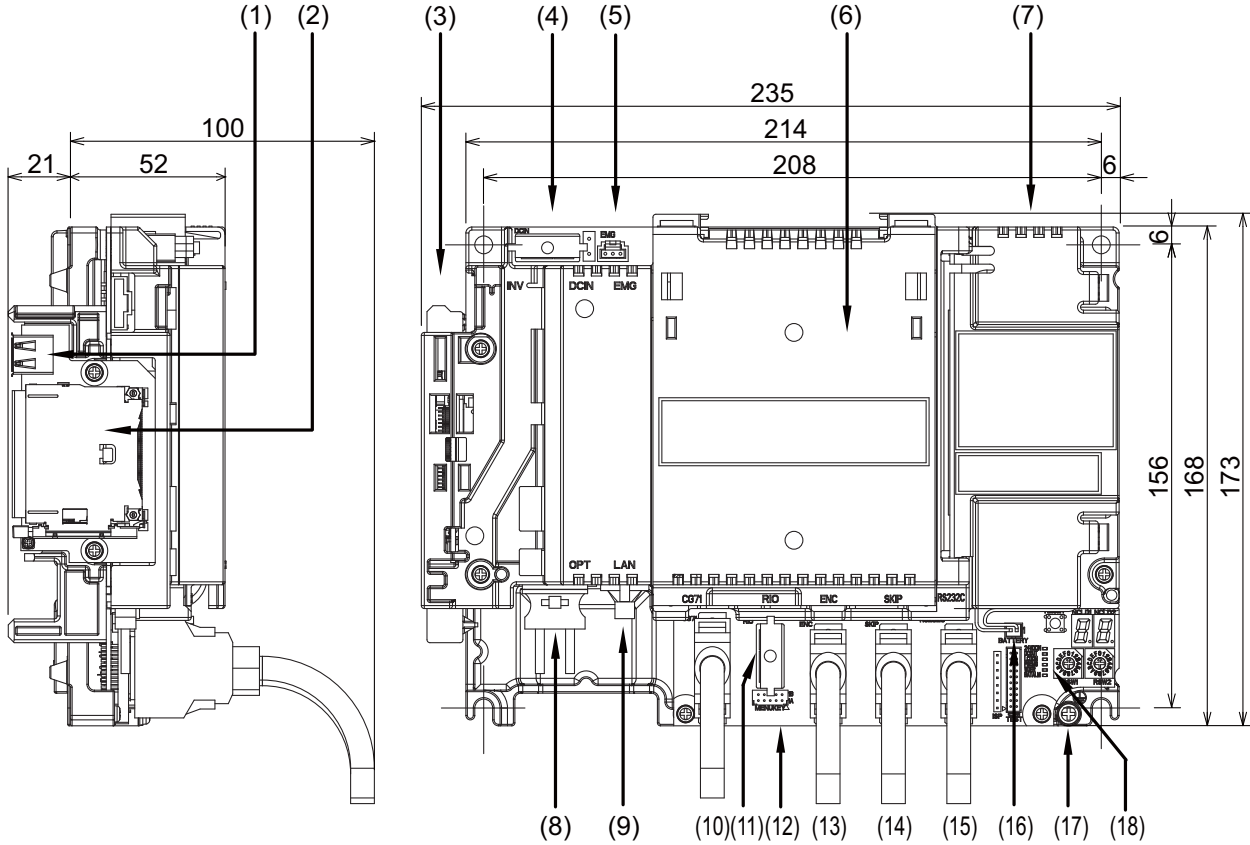
(Note 7) FCU7-KB926 does not need 24VDC power supply input.

Hardware Specifications

5.1 Control Unit

Dimension and names of parts

[FCU7-MU528(FCU7-MU558)]



No.	Connector name	Function	No.	Connector name	Function
(1)	USB	Front USB memory I/F	(11)	RIO1	Remote I/O unit I/F
(2)	CF	Front CF card I/F	(12)	MENUKEY	Menu key I/F
(3)	INV	Display unit backlight driver I/F	(13)	ENC	Encoder input 1ch (5V manual pulse generator input 2ch)
(4)	DCIN	24VDC input	(14)	SKIP	Skip input 8ch
(5)	EMG	External emergency stop input	(15)	SIO	Serial communication (RS-232C) I/F 2ch
(6)	ADONCCB	Expansion card slot	(16)	BAT	Battery (Q6BAT) I/F
(7)	LCD	Display unit signal I/F	(17)	FG	FG terminal
(8)	OPT	Optical communication I/F	(18)		LED
(9)	LAN	Ethernet I/F			
(10)	CG71	Operation panel I/O unit I/F			

(1) Front USB memory I/F (USB)

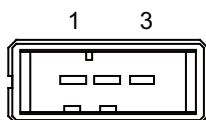
Do not connect devices other than USB memories.

When using a commercially available USB memory, performance check must be made by machine tool builder.

(2) Front CF card I/F (CF)

It is recommended to use CF cards of the original equipment manufactured parts.

Mitsubishi is unable to guarantee the machine operation when a commercially available CF card or SD memory card (SD-CF adapter is required) is used. In that case, performance check must be made carefully by machine tool builder.

(3) Display unit backlight driver I/F (INV)**(4) 24VDC input (DCIN)**

1	I	+24V
2		0V
3		FG

<Cable side connector type>

Connector: 2-178288-3

Contact: 1-175218-5 x3

Recommended manufacturer: Tyco Electronics

(a) Specifications of power supply

Consider the following characteristics when selecting the stabilized power supply (prepared by machine tool builder). Use a power supply that complies with CE Marking or that follows the safety standards given below.

[Stabilized power supply selection items]

Item		Standard setting
Output	Voltage fluctuation	±5% or less of 24VDC
	Ripple noise	200mV (P-P)
Power capacity		2.5A or more
Output holding time		20ms
Overcurrent protection		Required

[Standards]

Safety Standards: UL1950, CSA C22.2 No.234 approved, IEC950 compliant

Noise Terminal Voltage: FCC Class A, VCCI-Class A

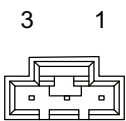
High Harmonics Current Restrictions: IEC61000-3-2

(Note) 24VDC voltage may drop instantaneously due to rush current at the beginning of 24V power supply to the control unit. The level of voltage drop depends on the capacity of the power supply. Do not share the power supply with the devices that have alarms to warn the voltage drop.


CAUTION

- Using a stabilized power supply without overcurrent protection may cause the unit's failure due to miswiring of 24V.

(5) External emergency stop input (EMG)



1		FG
2	I	EMG IN
3	O	+24V

* Be sure to connect EMG terminal cable (G123) to the connector when not used.

<Cable side connector type>

Connector: 005057-9403

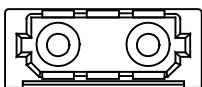
Contact: 0016020103

Recommended manufacturer: MOLEX

(6) Expansion card slot (ADONCCB)

(7) Display unit signal I/F (LCD)

(8) Optical communication I/F (OPT)



<Cable side connector type>

(PCF type)

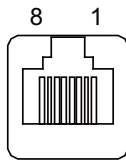
Connector: CF-2D101-S

Recommended manufacturer: Japan Aviation Electronics

(POF type)

Connector: PF-2D101

Recommended manufacturer: Japan Aviation Electronics

(9) Ethernet I/F (LAN)

1	O	TD+
2	O	TD-
3	I	RD+
4		
5		
6	I	RD-
7		
8		

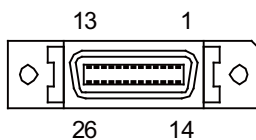
* Connect connector case with FG pattern.

* Use cross cable (G300) when directly connecting a device such as a personal computer to the unit.

<Cable side connector type>

Connector: 5-569550-3

Recommended manufacturer: Tyco Electronics

(10) Operation panel I/O unit I/F (CG71)

1		GND	14		GND
2		5V	15		5V
3		5V	16		3.3V
4		GND	17		GND
5	O	KBCS0*	18	O	KBCS1*
6	O	KBCS2*	19	O	KBAD0
7	O	KBAD1	20	O	KBAD2
8	I	KBD0	21	I	KBD1
9	I	KBD2	22	I	KBD3
10	O	KBRES*	23	O	RDYOUT*
11	O	BUZOUT*	24		3.3V
12	I/O	TXRX3	25	I/O	TXRX3*
13	O	SCAN36	26	O	SCAN37

* Connect connector case with FG pattern.

<Cable side connector type>

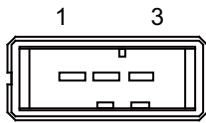
Plug: 10126-3000VE

Shell: 10326-52F0-008

Recommended manufacturer: 3M

(11) Remote I/O unit I/F (RIO1)

Up to eight remote I/O stations can be connected.



1	I/O	TXRX1
2	I/O	TXRX1*
3		0V

<Cable side connector type>

Connector: 1-178288-3

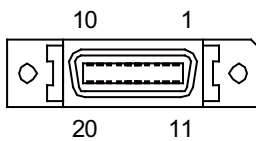
Contact: 1-175218-2 x3

Recommended manufacturer: Tyco Electronics

(12) Menu key I/F (MENUKEY)

(13) Encoder input 1ch/ 5V manual pulse generator input 2ch (ENC)

Synchronous feed encoder or 5V manual pulse generator can be connected to this connector.



1		0V	11		0V
2	I	ENC1Z	12	I	ENC1Z*
3	I	ENC1B	13	I	ENC1B*
4	I	ENC1A	14	I	ENC1A*
5		0V	15		0V
6	O	5V	16	O	5V
7	I	HA2A	17	I	HA2B
8	I	HA1A	18	I	HA1B
9		NC	19		NC
10	O	5V	20	O	5V

* Connect connector case with FG pattern.

<Cable side connector type>

Plug: 10120-3000VE

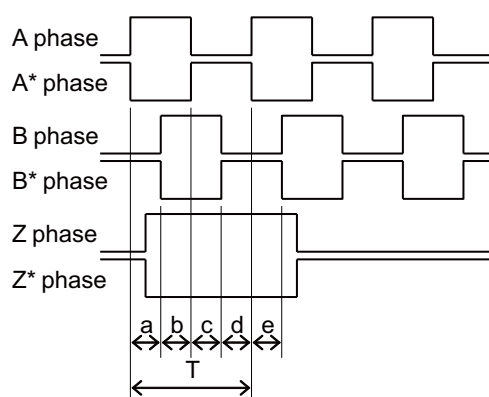
Shell: 10320-52F0-008

Recommended manufacturer: 3M

(a) Input for synchronous feed encoder

<Specification of input part>

Number of pulse phases		Three phases (A phase, B phase, a phase difference 90 degrees, Z phase) (Refer to the waveform below.)
Signal output of the encoder		Line driver output
Signal voltage	Input voltage range	0V to 5.25V
	Differential-input voltage VIT+	0.2V to 5.25V
	Differential-input voltage VIT-	-5.25V to -0.2V
Power supply voltage		5VDC±10%
Current consumption		200mA or less
Number of pulses per rotation		1024 pulse/rev
Input frequency (rotation speed)		136kHz or less (8000r/min or less)
Cable length		50m or less

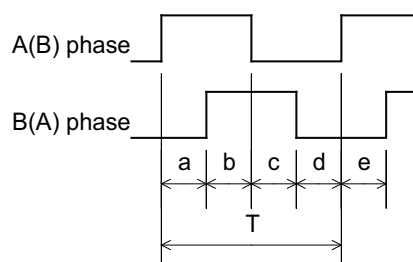


a.b.c.d.e: A phase or B phase rising edge (falling edge) phase difference = $T/4 \pm T/10$

(b) Input for 5V manual pulse generator

<Specification of input part>

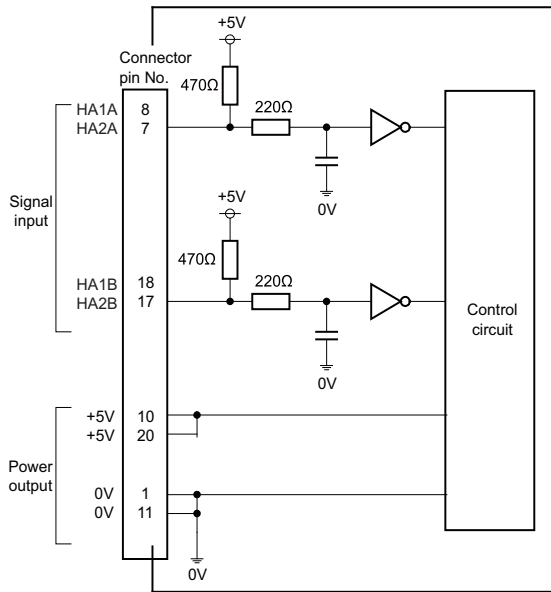
Number of pulse phases		Two phases (A phase, B phase, a phase difference 90 degrees) (Refer to the waveform below.)
Signal output of manual pulse generator		Voltage output, open collector output
Signal voltage		H level 3.5V to 5.25V L level 0V to 0.5V
Power supply voltage		5VDC±10%
Current consumption		100mA or less
Number of pulses per rotation		25 pulse/rev, 100 pulse/rev
Input frequency (rotation speed)		1kHz or less (40r/s or less for 25pulse/rev, 10r/s or less for 100pulse/rev)
Cable length		20m or less



a.b.c.d.e: A phase or B phase rising edge (falling edge) phase difference = $T/4 \pm T/10$

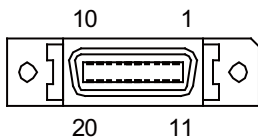
T: A or B phase cycle

(c) 5V manual pulse generator input/output circuit



When using the synchronous feed encoder and the manual pulse generator at the same time, connect the manual pulse generator to the operation panel I/O unit or use a distribution cable made by the machine tool builder.

(14) Skip input 8ch (SKIP)



1		COM	11		COM
2	I	SKIP0	12	I	SKIP1
3	I	SKIP2	13	I	SKIP3
4		NC	14		NC
5		COM	15		COM
6		NC	16		NC
7	I	SKIP4	17	I	SKIP5
8	I	SKIP6	18	I	SKIP7
9		NC	19		NC
10		NC	20		NC

* Connect connector case with FG pattern.

<Cable side connector type>

Plug: 10120-3000VE

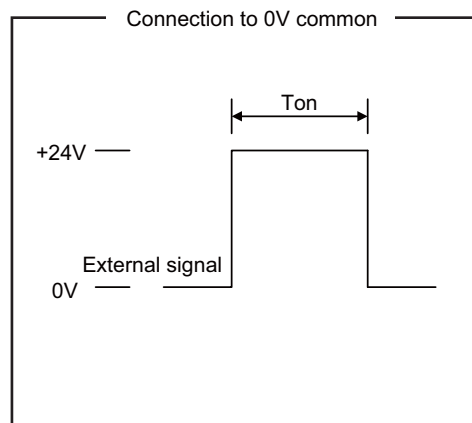
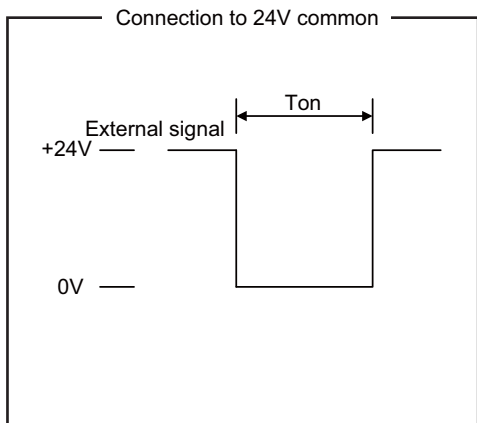
Shell: 10320-52F0-008

Recommended manufacturer: 3M

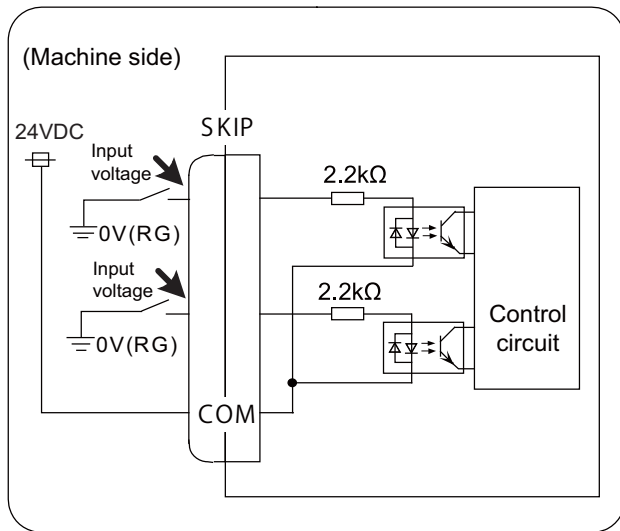
(a) Skip signal input conditions

Use the input signal within the following condition ranges.

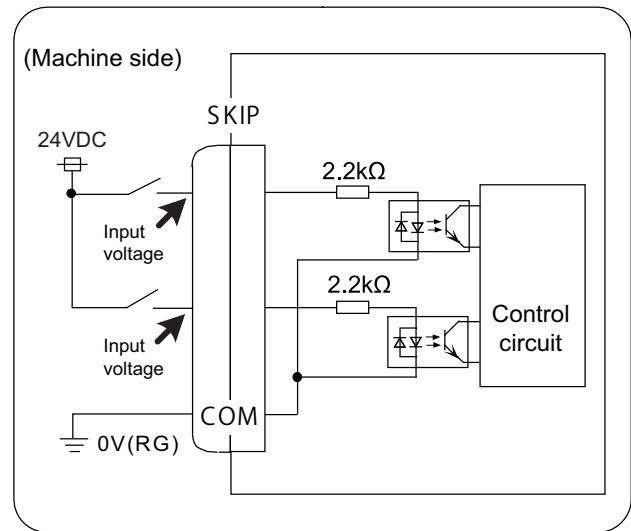
		24V common	0V common
1	Input voltage at external contact ON	6V or less	18V or more, 25.2V or less
2	Input current at external contact ON	6mA or more	
3	Input voltage at external contact OFF	20V or more, 25.2V or less	4V or less
4	Input current at external contact OFF	2mA or less	
5	Input resistance	Approx. 2.2kΩ	
6	Input signal holding time (Ton)	2ms or more	
7	Internal response time	0.08ms or less	
8	Machine side contact capacity	+30V or more, 16mA or more	



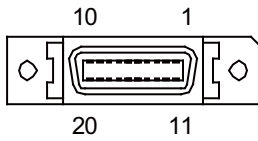
Connection to 24V common



Connection to 0V common



(15) Serial communication (RS-232C) I/F 2ch (SIO)



1		0V	11		0V
2	I	RD1(RXD1)	12	O	SD1(TXD1)
3	I	CS1(CTS1)	13	O	RS1(RTS1)
4	O	DR1(DSR1)	14	I	ER1(DTR1)
5		0V	15		0V
6		NC	16		NC
7	I	RD2(RXD2)	17	O	SD2(TXD2)
8	I	CS2(CTS2)	18	O	RS2(RTS2)
9	O	DR2(DSR2)	19	I	ER2(DTR2)
10		NC	20		NC

* Connect connector case with FG pattern.

<Cable side connector type>

Plug: 10120-3000VE

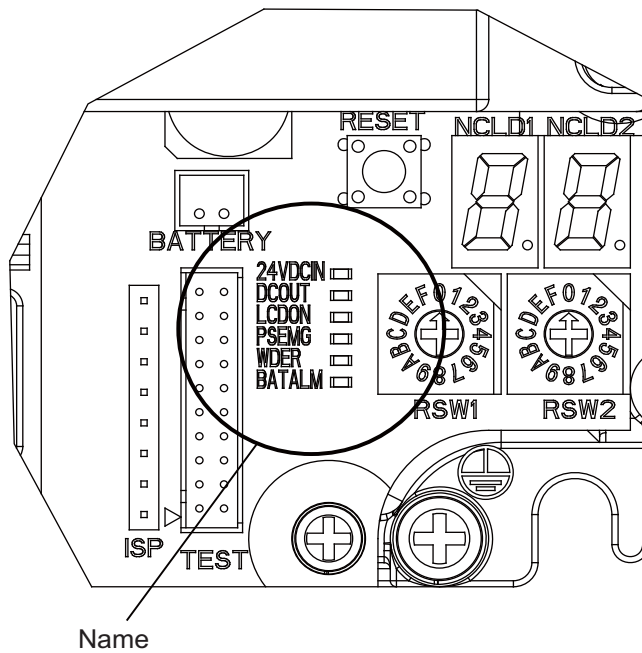
Shell: 10320-52F0-008

Recommended manufacturer: 3M

(16) Battery (Q6BAT) I/F (BAT)

(17) FG terminal (FG)

(18) LED

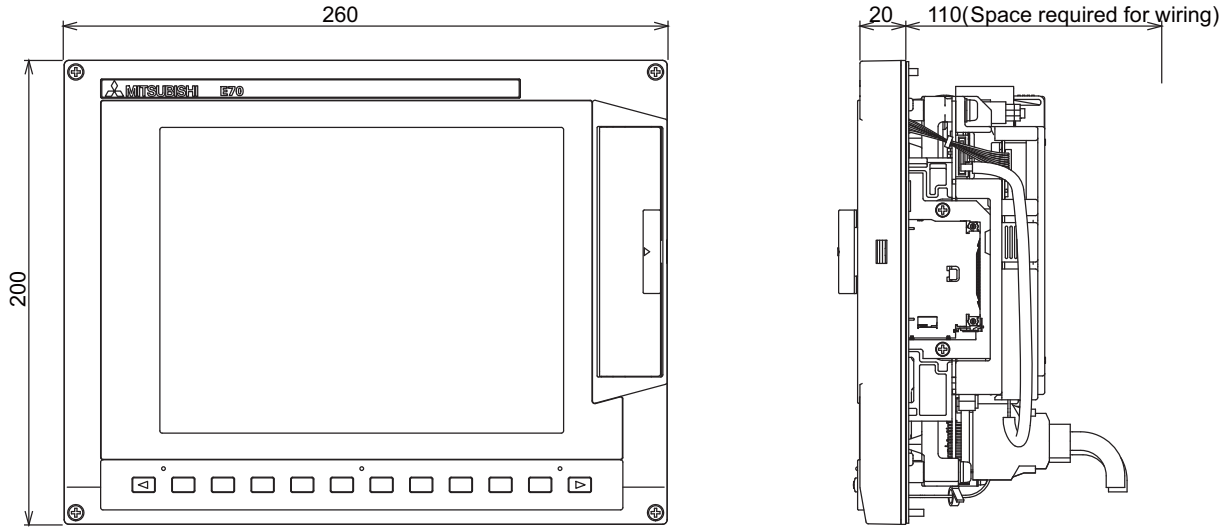


Name	Function	At fault	Conditions
24VDCIN	24VDC input check	Not lit	(1) Failure of 24VDC input (2) Fuse is disconnected near DCIN connector
DCOUT	Internal output voltage check	Not lit	(1) Failure of internal voltage output in control unit (2) Short circuit of 5VDC output on CG71 or ENC
LCDON	12VDC output voltage check for backlight driver	Not lit	(1) Failure of 12VDC output in control unit (2) 24VDC input voltage is +20V or less
PSEMG	External emergency stop status display	Lit (Red)	External emergency stop signal has inputted
WDER	System error display	Lit (Red)	(1) Failure of control unit (2) SRAM data is broken
BATALM	Battery voltage drop	Lit (Red)	Battery voltage has dropped to 2.7V or less

5.2 Display Unit

Outline dimension

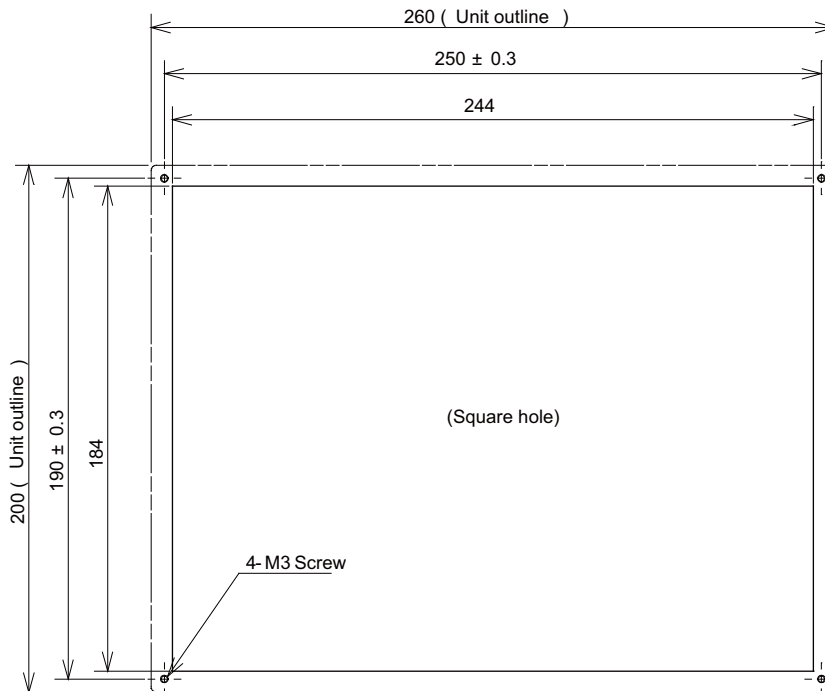
[FCU7-DU120-13 (8.4-type)]



(Note 1) The above side view shows the state with the control unit mounted.

(Note 2) Consider the minimum radius value of optical communication cable for the bottom space.

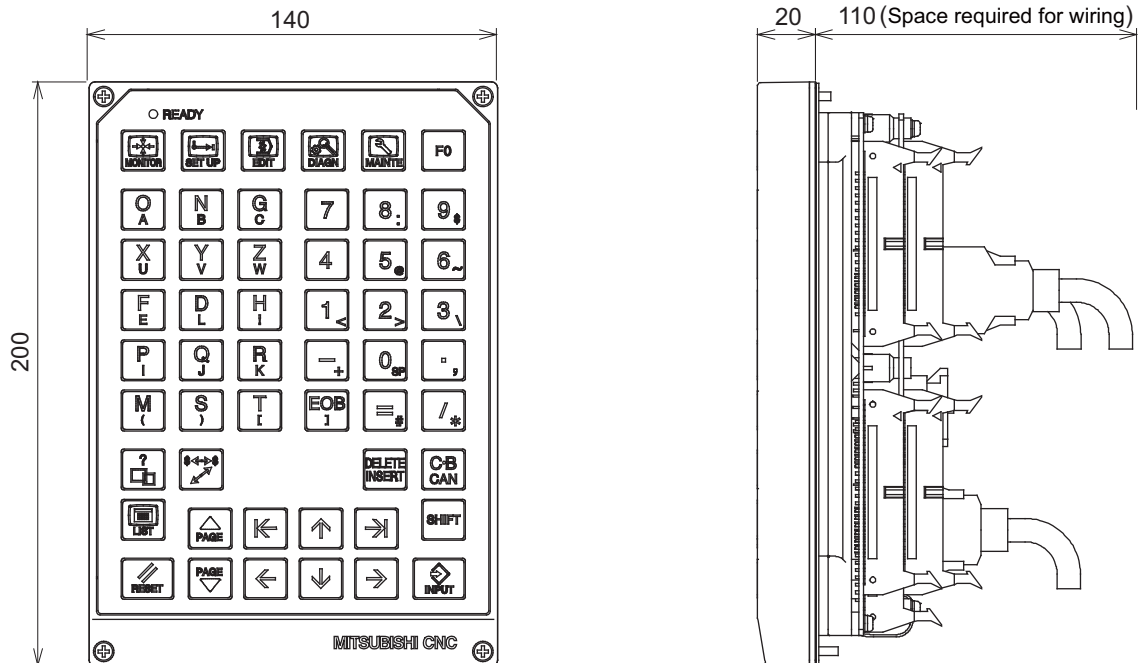
<Panel cut dimension drawing>



5.3 Keyboard Unit

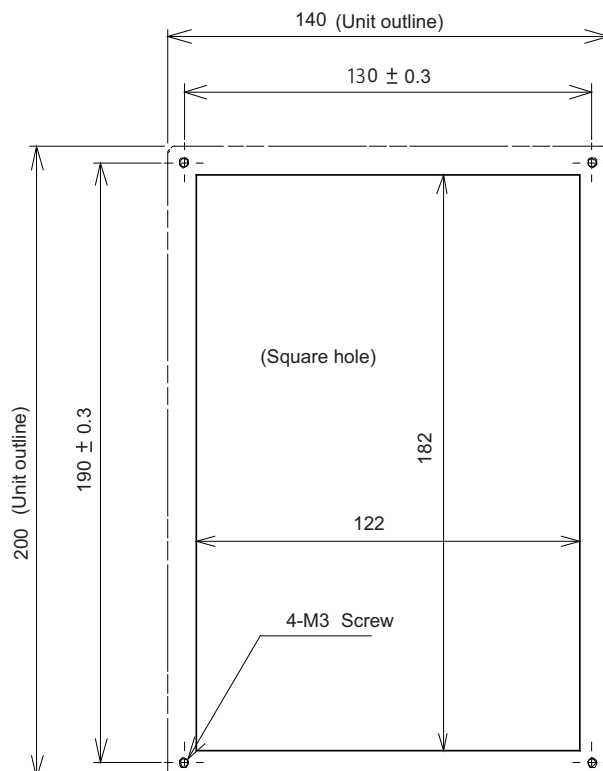
Outline dimension

[FCU7-KB024 (8.4-type)]

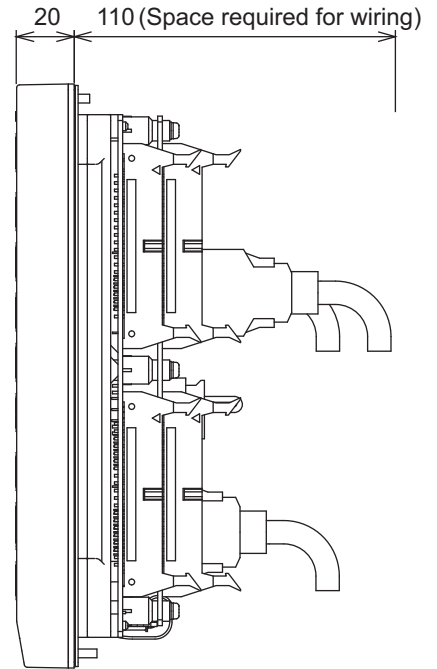
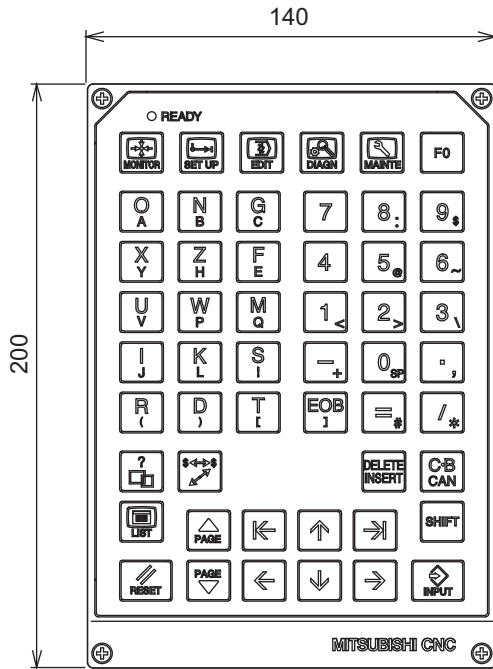


(Note) The above side view shows the state with the operation panel I/O unit FCU7-DX721 mounted.

<Panel cut dimension drawing>

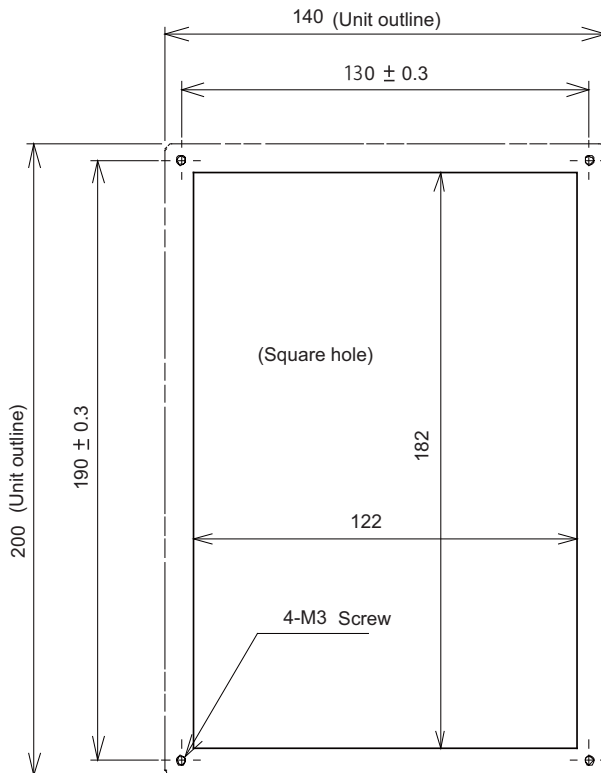


[FCU7-KB025 (8.4-type)]



(Note) The above side view shows the state with the operation panel I/O unit FCU7-DX721 mounted.

<Panel cut dimension drawing>



5.4 Operation Panel I/O Unit

Characteristics of operation panel I/O unit are as follows.

- (1) Number of DI/DO points that can be mounted on the machine operation panel is 64/64 at the minimum and 96/96 at the maximum.
Operation panel I/O unit DI/DO uses equivalent serial link connections as those used for remote I/O.
- (2) Remote I/O interface --- 1ch
Extension with remote I/O unit is possible up to 4 stations.
(a) When FCU7-DX711 is used: Up to 4 stations, 128 points/128 points in total, are available.
(b) When any one of FCU7-DX621/721/731 is used: Up to 3 stations, 96 points/96 points in total, are available.
(Note) The maximum number of stations that can be extended is described above whether DI/DO of the operation panel I/O unit is used or not.
- (3) Manual pulse generator --- 2ch
5V and 12V manual pulse generators can be connected.
- (4) Installation on the back side of the keyboard unit is possible. Allows space saving inside the operation panel.

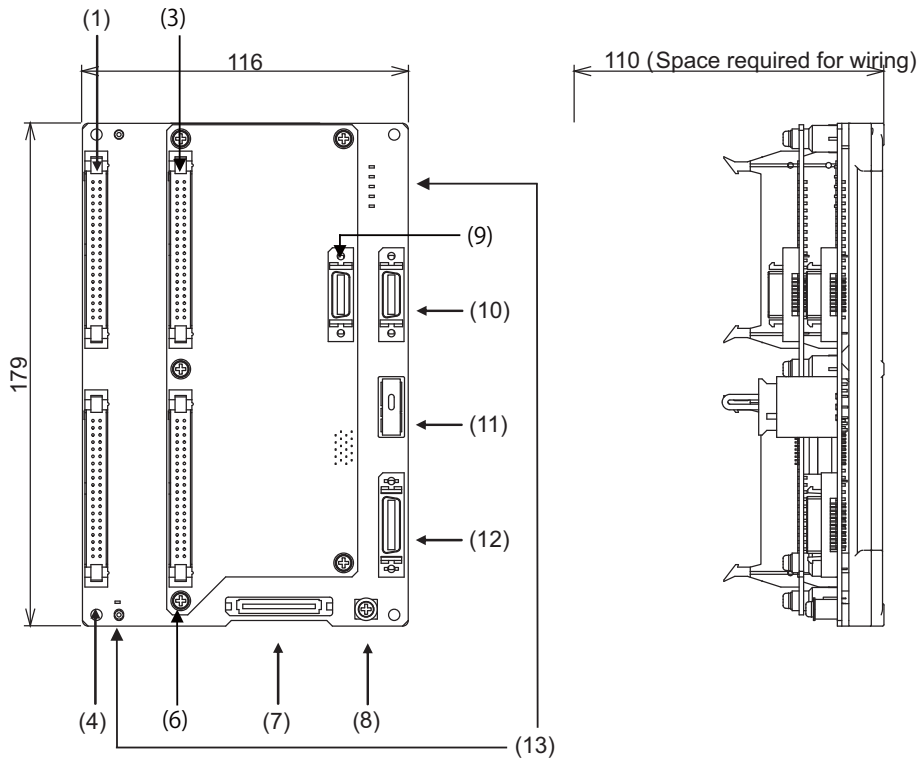
Classification	Type	Components	Remarks
DI 24V/0V common input DO Source output AO Analog output	FCU7-DX621	Base card Terminator (R-TM) Add-on card	DI: 64-points 24V/0V common type DO: 48-points source type MPG:2ch AO: 1 point Occupied stations (fixed): 1, 2, 3, 7, 8 RIO3 extensible stations: 4, 5, 6
DI 24V/0V common input DO Source output	FCU7-DX711	Base card Terminator (R-TM)	DI: 64-points 24V/0V common type DO: 64-points source type MPG:2ch Occupied stations (fixed): 1, 2, 7, 8 RIO3 extensible stations: 3, 4, 5, 6
DI 24V/0V common input DO Source output AO Analog output	FCU7-DX721	Base card Terminator (R-TM) Add-on card	DI: 96-points 24V/0V common type DO: 80-points source type MPG:2ch AO: 1 point Occupied stations (fixed): 1, 2, 3, 7, 8 RIO3 extensible stations: 4, 5, 6
DI 24V/0V common input DO Source output	FCU7-DX731	Base card Terminator (R-TM) Add-on card	DI: 96-points 24V/0V common type DO: 96-points source type MPG:2ch Occupied stations (fixed): 1, 2, 3, 7, 8 RIO3 extensible stations: 4, 5, 6

(Note 1) The station Nos. occupied by the operation panel I/O unit cannot be changed.

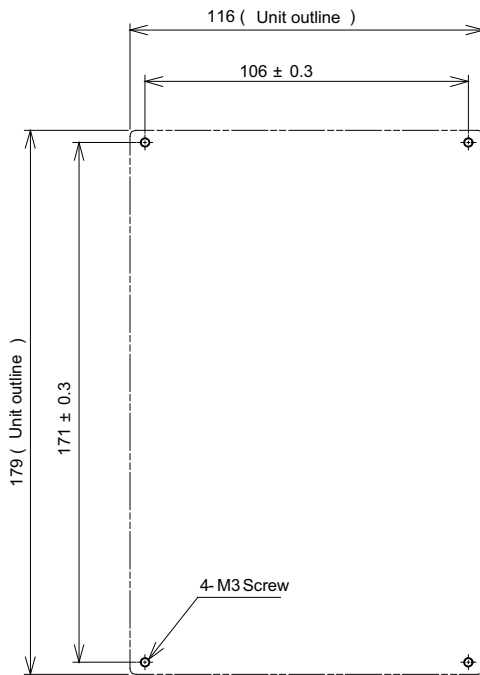
(Note 2) Set the number of DI points that are simultaneously turned ON to be less than half of the total points. If many points are set to be simultaneously turned ON in high temperature, operation panel I/O unit may be deteriorated due to the heat.

Dimension and names of parts

[FCU7-DX621]

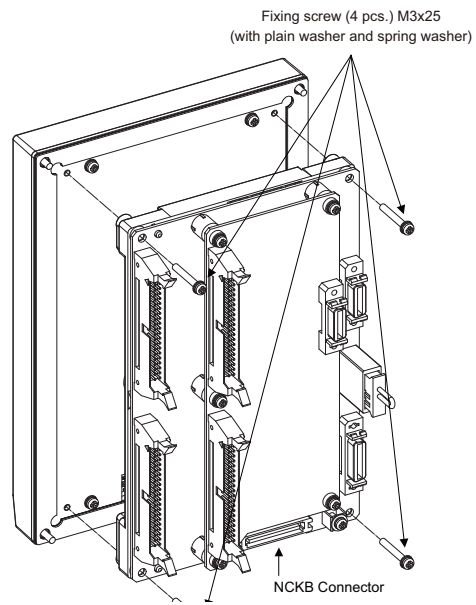


<Dimension drawing for installing on the panel>



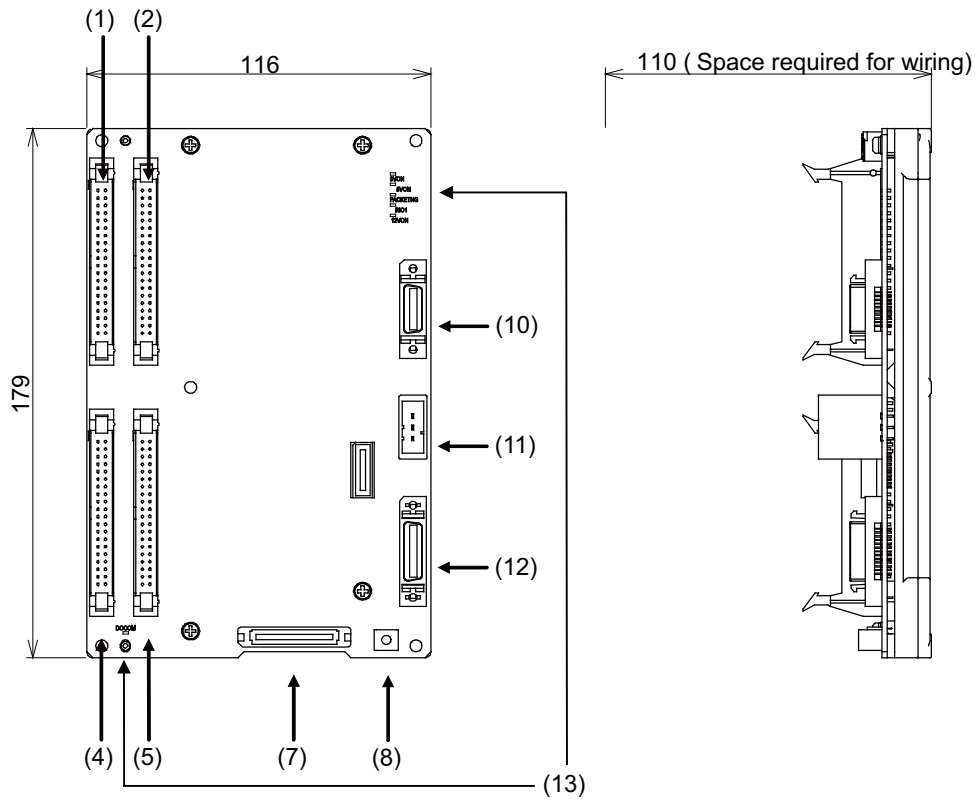
<Installation on the back side of the keyboard>

Operation panel I/O unit can be installed on the back side of the keyboard.

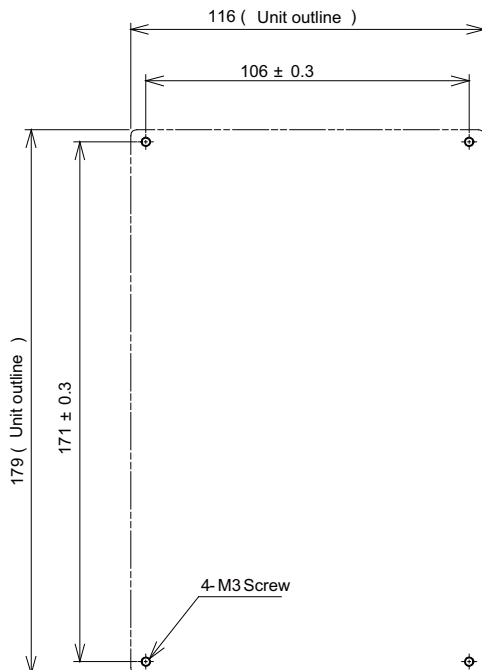


(Note) Use M3x25 screws (with spring washer and plain washer) when mounting the operation panel I/O unit.

[FCU7-DX711]

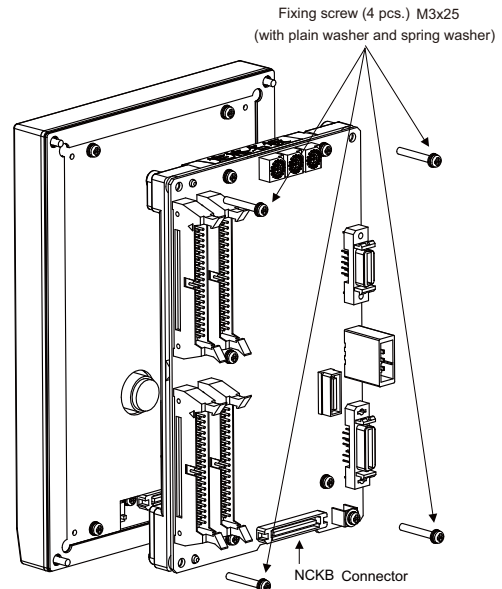


<Dimension drawing for installing on the panel>



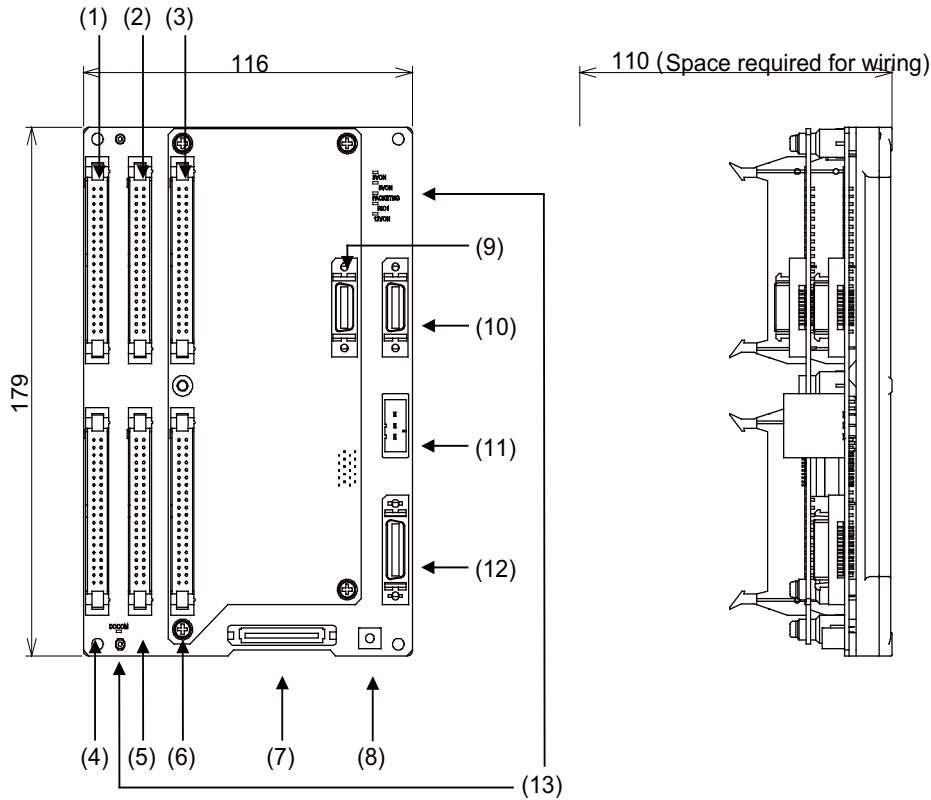
<Installation on the back side of the keyboard>

Operation panel I/O unit can be installed on the back side of the keyboard.



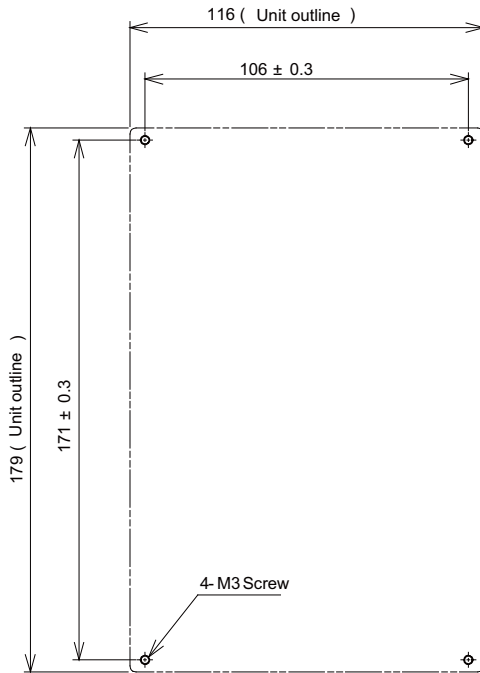
(Note) Use M3x25 screws (with spring washer and plain washer) when mounting the operation panel I/O unit.

[FCU7-DX721/ FCU7-DX731]



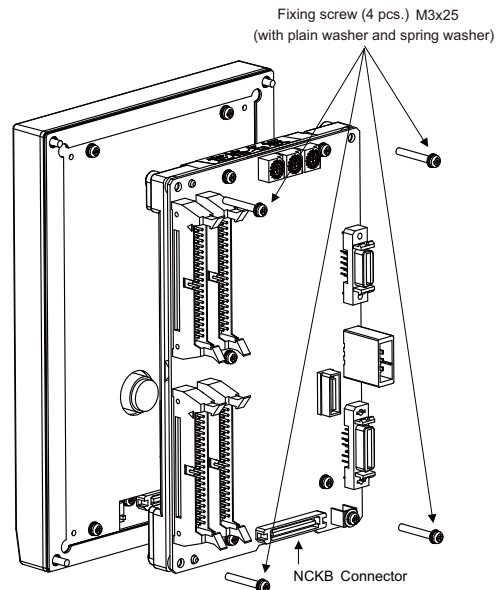
(Note) (9) is not available for FCU7-DX731.

<Dimension drawing for installing on the panel>



<Installation on the back side of the keyboard>

Operation panel I/O unit can be installed on the back side of the keyboard.



(Note) Use M3x25 screws (with spring washer and plain washer) when mounting the operation panel I/O unit.

(1) Machine input (CG31)

Digital input 32 points (1st station)

(2) Machine input (CG33)

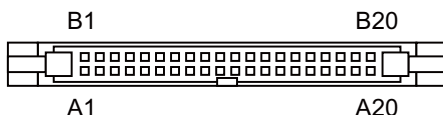
Digital input 32 points (2nd station)

(Note) This is not available for FCU7-DX621.

(3) Machine input (CG35)

Digital input 32 points (3rd station)

(Note) This is not available for FCU7-DX711.



<Cable side connector type>

Connector: 7940-6500SC

Strain relief: 3448-7940

Recommended manufacturer: 3M

CG31						CG33						CG35					
B			A			B			A			B			A		
20	I	X200	20	I	X210	20	I	X220	20	I	X230	20	I	X240	20	I	X250
19	I	X201	19	I	X211	19	I	X221	19	I	X231	19	I	X241	19	I	X251
18	I	X202	18	I	X212	18	I	X222	18	I	X232	18	I	X242	18	I	X252
17	I	X203	17	I	X213	17	I	X223	17	I	X233	17	I	X243	17	I	X253
16	I	X204	16	I	X214	16	I	X224	16	I	X234	16	I	X244	16	I	X254
15	I	X205	15	I	X215	15	I	X225	15	I	X235	15	I	X245	15	I	X255
14	I	X206	14	I	X216	14	I	X226	14	I	X236	14	I	X246	14	I	X256
13	I	X207	13	I	X217	13	I	X227	13	I	X237	13	I	X247	13	I	X257
12	I	X208	12	I	X218	12	I	X228	12	I	X238	12	I	X248	12	I	X258
11	I	X209	11	I	X219	11	I	X229	11	I	X239	11	I	X249	11	I	X259
10	I	X20A	10	I	X21A	10	I	X22A	10	I	X23A	10	I	X24A	10	I	X25A
9	I	X20B	9	I	X21B	9	I	X22B	9	I	X23B	9	I	X24B	9	I	X25B
8	I	X20C	8	I	X21C	8	I	X22C	8	I	X23C	8	I	X24C	8	I	X25C
7	I	X20D	7	I	X21D	7	I	X22D	7	I	X23D	7	I	X24D	7	I	X25D
6	I	X20E	6	I	X21E	6	I	X22E	6	I	X23E	6	I	X24E	6	I	X25E
5	I	X20F	5	I	X21F	5	I	X22F	5	I	X23F	5	I	X24F	5	I	X25F
4		NC	4		NC	4		NC	4		NC	4		NC	4		NC
3		COM	3		COM	3		COM	3		COM	3		COM	3		COM
2	I	+24V	2		0V	2	I	+24V	2		0V	2	I	+24V	2		0V
1	I	+24V	1		0V	1	I	+24V	1		0V	1	I	+24V	1		0V

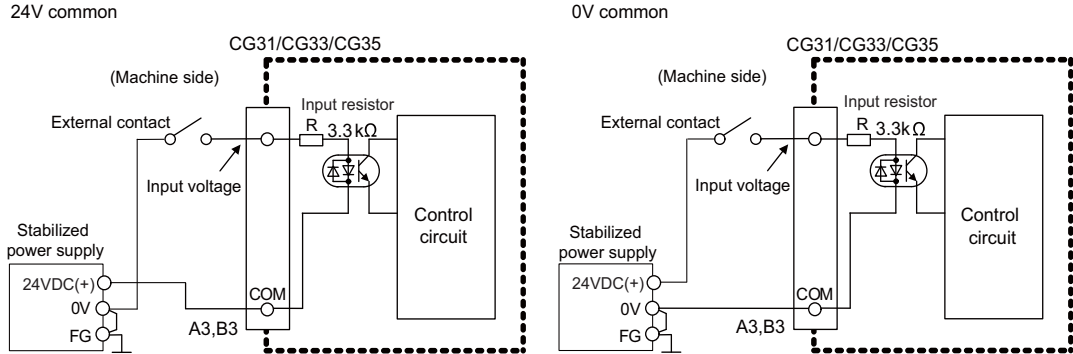
Each station has fixed input/output assignments. Refer to "PLC Interface Manual" for details.

(a) Outline of digital signal input circuit

Both 24V common and 0V common connections are allowed in the digital signal input circuit.

Follow the wiring diagram below for each type.

Input circuit

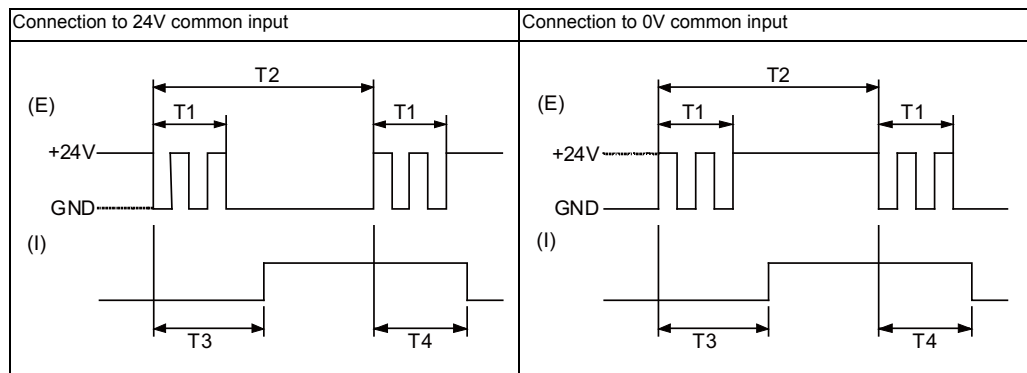


Input conditions

The input signals must be used within the following condition ranges.

		24V common	0V common
1	Input voltage at external contact ON	6V or less	18V or more, 25.2V or less
2	Input current at external contact ON	9mA or more	
3	Input voltage at external contact OFF	20V or more, 25.2V or less	4V or less
4	Input current at external contact OFF	2mA or less	
5	Input resistance	Approx. 3.3kΩ	
6	Tolerable chattering time (T1)	3ms	
7	Input signal holding time (T2)	40ms or more (Note)	
8	input circuit operation delay time (T3 and T4)	3 to 16ms	
9	Machine side contact capacity	30V or more, 16mA or more	

(Note) Input signal holding time: The guide is 40ms or more. The input signal will not be recognized unless it is held for the ladder processing cycle time or longer.



(E) : External signal, (I):Internal signal

(4) Machine output (CG32)

Digital output 32 points (1st station)

(5) Machine output (CG34)

Digital output 32 points (2nd station)

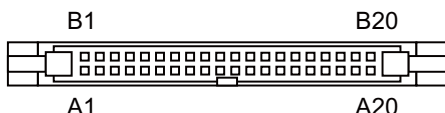
(Note) This is not available for FCU7-DX621.

(6) Machine output (CG36)

FCU7-DX731: Digital output 32 points (3rd station)

FCU7-DX621/DX721: Digital output 16 points (3rd station) (with analog output)

(Note) This is not available for FCU7-DX711.



<Cable side connector type>

Connector: 7940-6500SC

Strain relief: 3448-7940

Recommended manufacturer: 3M

CG32						CG34						CG36 (Note 1)					
B			A			B			A			B			A		
20	O	Y200	20	O	Y210	20	O	Y220	20	O	Y230	20	O	Y240	20	O	Y250
19	O	Y201	19	O	Y211	19	O	Y221	19	O	Y231	19	O	Y241	19	O	Y251
18	O	Y202	18	O	Y212	18	O	Y222	18	O	Y232	18	O	Y242	18	O	Y252
17	O	Y203	17	O	Y213	17	O	Y223	17	O	Y233	17	O	Y243	17	O	Y253
16	O	Y204	16	O	Y214	16	O	Y224	16	O	Y234	16	O	Y244	16	O	Y254
15	O	Y205	15	O	Y215	15	O	Y225	15	O	Y235	15	O	Y245	15	O	Y255
14	O	Y206	14	O	Y216	14	O	Y226	14	O	Y236	14	O	Y246	14	O	Y256
13	O	Y207	13	O	Y217	13	O	Y227	13	O	Y237	13	O	Y247	13	O	Y257
12	O	Y208	12	O	Y218	12	O	Y228	12	O	Y238	12	O	Y248	12	O	Y258
11	O	Y209	11	O	Y219	11	O	Y229	11	O	Y239	11	O	Y249	11	O	Y259
10	O	Y20A	10	O	Y21A	10	O	Y22A	10	O	Y23A	10	O	Y24A	10	O	Y25A
9	O	Y20B	9	O	Y21B	9	O	Y22B	9	O	Y23B	9	O	Y24B	9	O	Y25B
8	O	Y20C	8	O	Y21C	8	O	Y22C	8	O	Y23C	8	O	Y24C	8	O	Y25C
7	O	Y20D	7	O	Y21D	7	O	Y22D	7	O	Y23D	7	O	Y24D	7	O	Y25D
6	O	Y20E	6	O	Y21E	6	O	Y22E	6	O	Y23E	6	O	Y24E	6	O	Y25E
5	O	Y20F	5	O	Y21F	5	O	Y22F	5	O	Y23F	5	O	Y24F	5	O	Y25F
4		COM (Note 2)	4		COM (Note 2)	4		COM (Note 2)	4		COM (Note 2)	4		COM (Note 2)	4		COM (Note 2)
3		COM (Note 2)	3		COM (Note 2)	3		COM (Note 2)	3		COM (Note 2)	3		COM (Note 2)	3		COM (Note 2)
2	I	24VDC	2		GND (0V)	2	I	24VDC	2		GND (0V)	2	I	24VDC	2		GND (0V)
1	I	24VDC	1		GND (0V)	1	I	24VDC	1		GND (0V)	1	I	24VDC	1		GND (0V)

(Note 1) FCU7-DX621/DX721 uses 16 points of digital outputs, Y240 to Y24F.

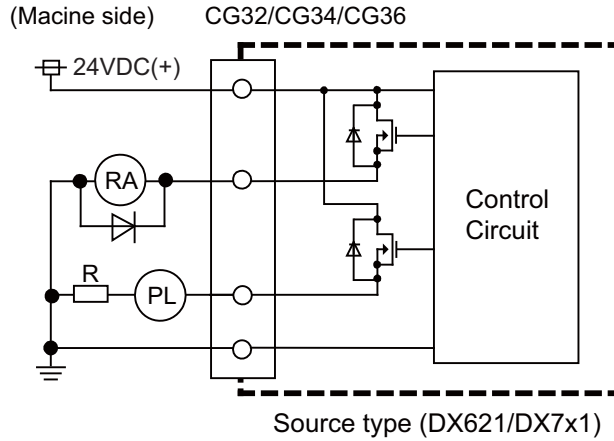
(Note 2) COM (3A, 3B, 4A, and 4B) is connected to 24VDC inside the unit.

Each station has fixed input/output assignments. Refer to "PLC Interface Manual" for details.

(a) Outline of digital signal output circuit

Use within the specification ranges shown below.

Output circuit



RA : Relay
PL : Pilot lamp

Output conditions

Insulation method	Non-insulation
Rated load voltage	24VDC
Max. output current	60mA/point
Output delay time	40μs

(Note 1) When using an inductive load such as a relay, always connect a diode (voltage resistance 100V or more, 100mA or more) in parallel to the load.

(Note 2) When using a capacitive load such as a lamp, always connect a protective resistor (R=150Ω) serially to the load to suppress rush currents. (Make sure that the current is less than the above tolerable current including the momentary current.)

⚠ CAUTION

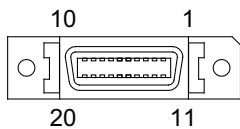
1. When using an inductive load such as a relay, always connect a diode in parallel to the load.
2. When using a capacitive load such as a lamp, always connect a protective resistor serially to the load to suppress rush currents.

(7) Keyboard I/F (NCKB)

(8) FG terminal (FG)

(9) Analog output 1ch (AO)

(Note) This is not available for FCU7-DX731.



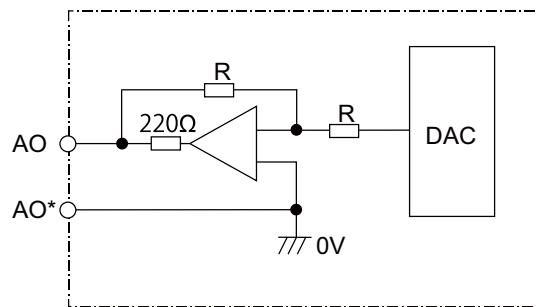
1		AO*	11		NC
2		NC	12		NC
3		NC	13		NC
4		NC	14		NC
5		NC	15		NC
6		NC	16		NC
7	○	AO	17		NC
8		NC	18		NC
9		NC	19		NC
10		NC	20		NC

* Connect connector case with FG pattern.

<Cable side connector type>

Plug: 10120-3000VE
Shell: 10320-52F0-008
Recommended manufacturer: 3M

Output circuit



Output conditions

Output voltage	0V to ±10V (±5%)
Resolution	12bit (±10V × n/4096) (Note)
Load conditions	10kΩ load resistance
Output impedance	220Ω

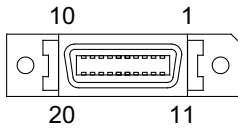
(Note) $n=(2^0 \text{ to } 2^{11})$

Connector pin assignment

1	AO*	GND
7	AO	Analog voltage output

(10) Manual pulse generator input 2ch (5V and 12V) (MPG)

5V manual pulse generator and 12V manual pulse generator can be connected to this connector.
Connect the synchronous feed encoder to the ENC connector of the control unit.



1		GND	11		GND
2		reserve	12		reserve
3	I	HA2A	13	I	HA2B
4	I	HA1A	14	I	HA1B
5		GND	15		GND
6	O	12VDC	16	O	12VDC
7		reserve	17		reserve
8		reserve	18		reserve
9		reserve	19		reserve
10	O	5VDC	20	O	5VDC

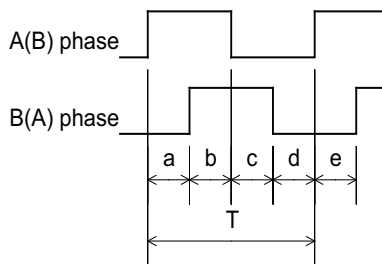
* Connect connector case with FG pattern.

<Cable side connector type>

Plug: 10120-3000VE
Shell: 10320-52F0-008
Recommended manufacturer: 3M

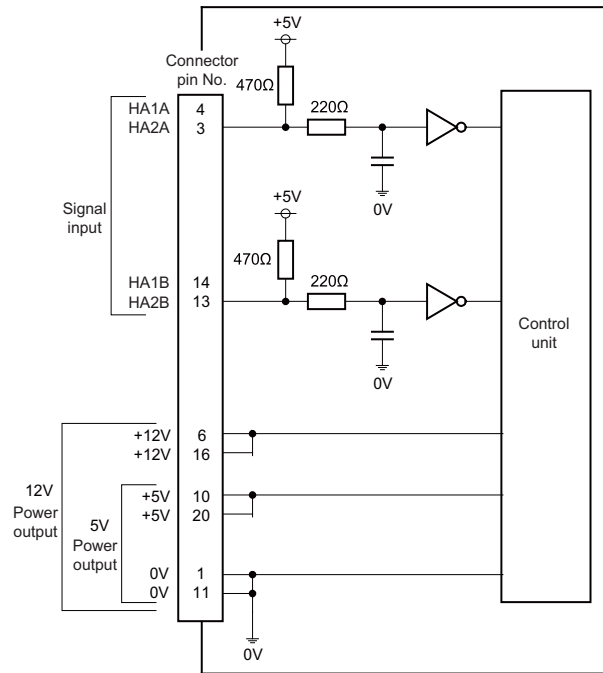
<Specification of input part>

	5V manual pulse generator	12V manual pulse generator
Number of pulse phases	Two phases (A phase, B phase, a phase difference 90 degrees) (Refer to the waveform below.)	
Signal output of manual pulse generator	Voltage output, open collector output	Open collector output
Signal voltage	H level 3.5V to 5.25V L level 0V to 0.5V	
Power supply voltage	5VDC±10%	12VDC±10%
Current consumption	100mA or less	
Number of pulses per rotation	25 pulse/rev, 100 pulse/rev	
Input frequency (rotation speed)	1kHz or less (40r/s or less for 25pulse/rev, 10r/s or less for 100pulse/rev)	
Cable length	20m or less	50m or less

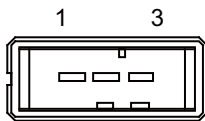


a.b.c.d.e: A phase or B phase rising edge (falling edge) phase difference = $T/4 \pm T/10$
T: A or B phase cycle

<Input/output circuit>



(11) Remote I/O unit I/F (RIO3)



1	I/O	TXRX3
2	I/O	TXRX3*
3		0V(GND)

<Cable side connector type>

Connector: 1-178288-3

Contact: 1-175218-2 x3

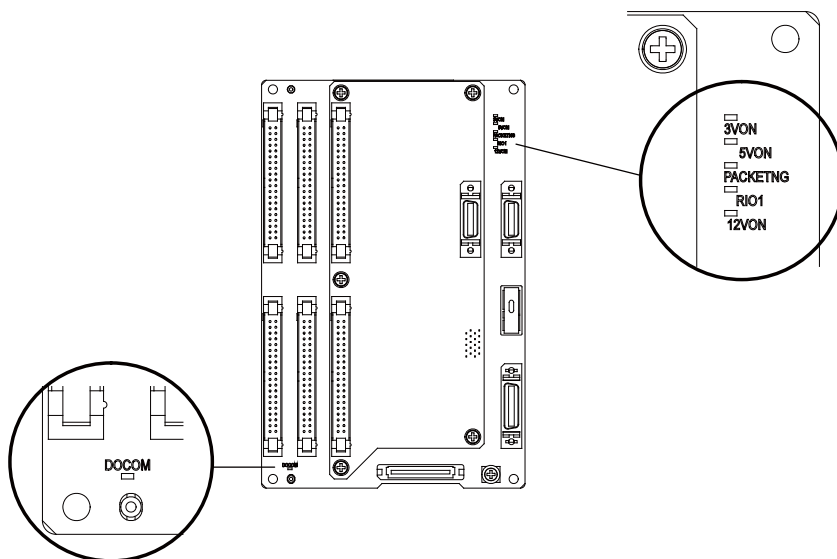
Recommended manufacturer: Tyco Electronics

Refer to the following chart for the maximum number of connecting stations and I/O points.

Operation panel I/O unit type	Max. number of stations (RIO3 connection)	Max. number of I/O points (RIO3 connection)
FCU7-DX711	4 stations (stations 3rd to 6th can be used)	128 points/128 points
FCU7-DX621/DX721/DX731	3 stations (stations 4th to 6th can be used)	96 points/96 points

(12) Control unit I/F (CG71)

(13) LED



Name	Function	At fault	Conditions
3VON	3VDC input check	Not lit	(1) Disconnection between control unit and operation panel I/O unit
5VON	5VDC input check	Not lit	(1) Disconnection between control unit and operation panel I/O unit (2) Short circuit of manual pulse generator cable
PACKETNG	RIO communication status check	Lit (Red)	(1) Partial disconnection between control unit and operation panel I/O unit (2) Miswiring of extended I/O unit to RIO3
RIO1	RIO communication status check	Not lit	
12VON	For manual pulse generator	Not lit	Short circuit of manual pulse generator cable
DOCOM	24VDC input check	Not lit	(1) Disconnection of 24V input (2) Disconnection of protection fuse on the operation panel I/O unit

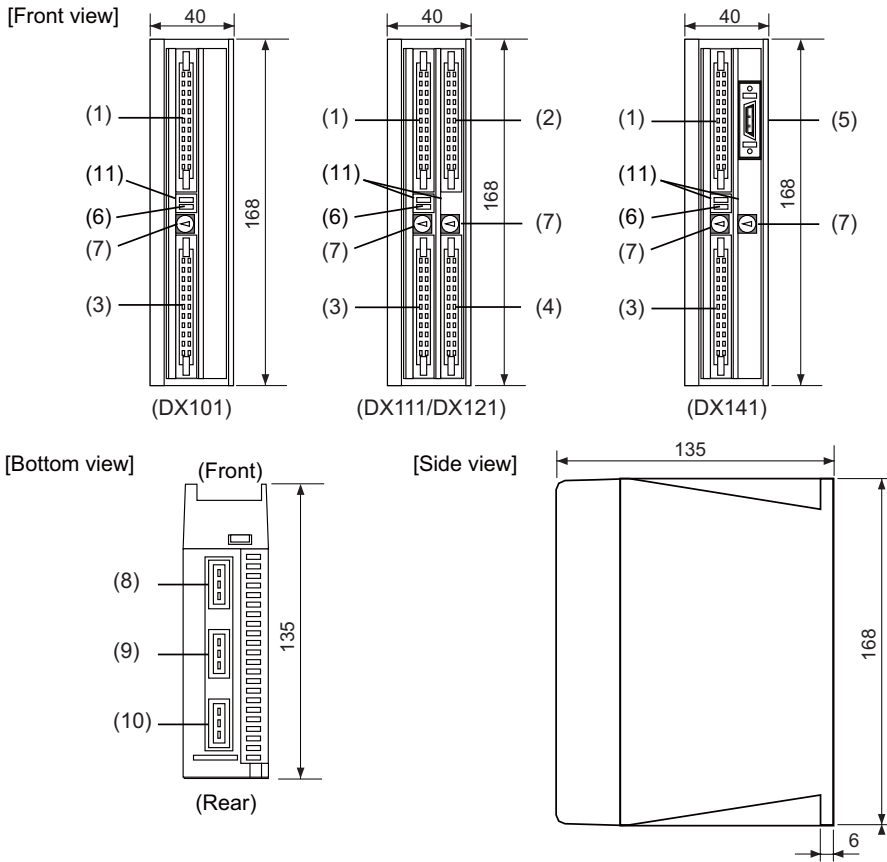
5.5 Remote I/O Unit

The following four types of signals can be input/output from the remote I/O unit (FCUA-DX1x1) according to the type and No. of contacts. Use serial link connections to connect the unit with the control unit or the operation panel I/O unit.

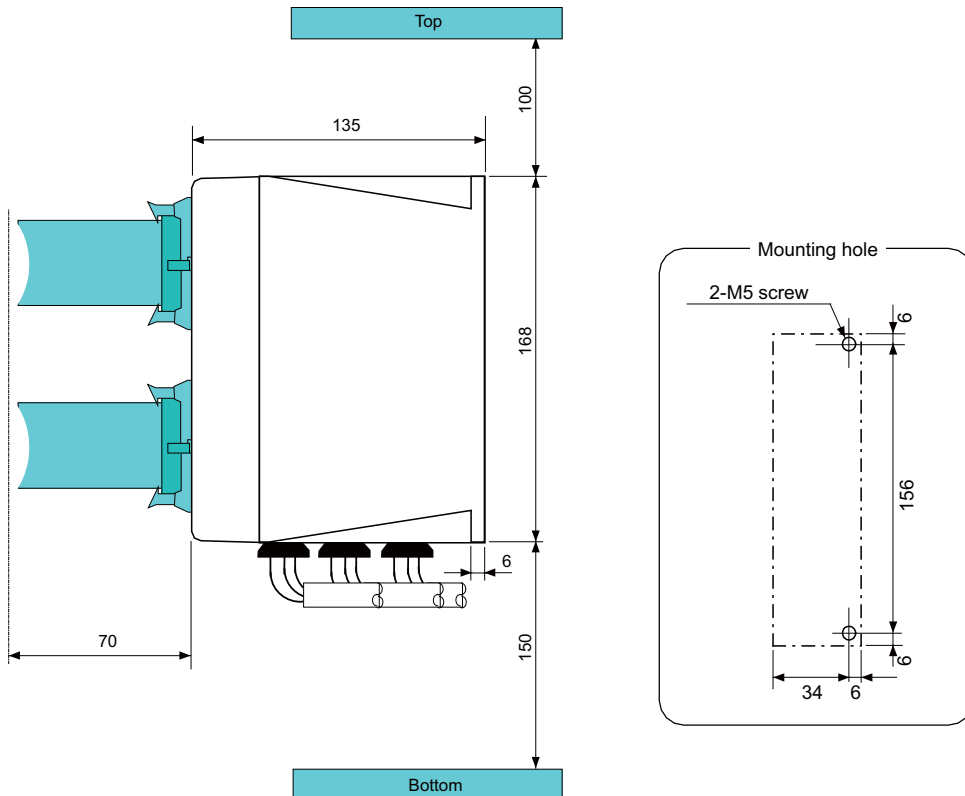
Multiple remote I/O units can be used as long as the total number of occupied stations is eight or less.

Classification	Type	Components	Remarks
DI 24V/0V common input DO Source output	FCUA-DX101	RX312	DI: 32-points 24V/0V common type (photo coupler insulation) DO: 32-points source type (non-insulation) Number of occupied stations: 1
DI 24V/0V common input DO Source output	FCUA-DX111	RX312+RX322-1	DI: 64-points 24V/0V common type (photo coupler insulation) DO: 48-points source type (non-insulation) Number of occupied stations: 2
DI 24V/0V common input DO Source output AO Analog output	FCUA-DX121	RX312+RX322	DI: 64-points 24V/0V common type (photo coupler insulation) DO: 48-points source type (non-insulation) AO: 1 point Number of occupied stations: 2
DI 24V/0V common input DO Source output AI Analog input AO Analog output	FCUA-DX141	RX312+RX341	DI: 32-points 24V/0V common type (photo coupler insulation) DO: 32-points source type (non-insulation) AI: 4 points AO: 1 point Number of occupied stations: 2

Dimension and names of parts



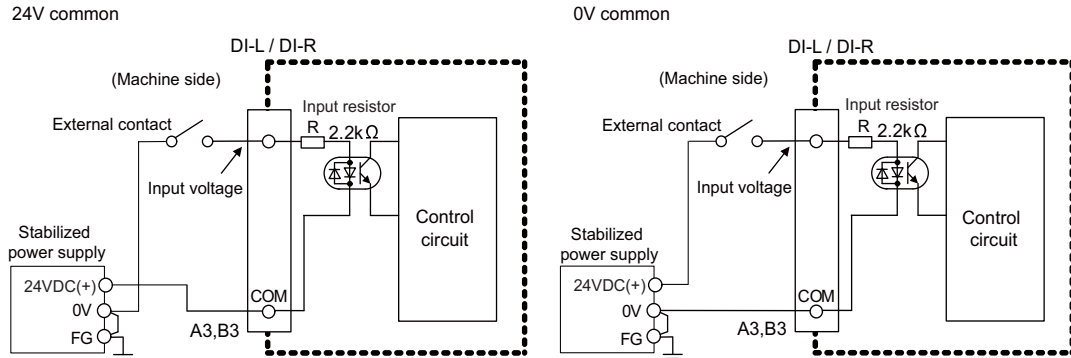
<Installation dimension drawing>



- (1) Machine input (DI-L)
- (2) Machine input (DI-R)

Both 24V common and 0V common connections are allowed in the digital signal input circuit.
Follow the wiring diagram below for each type.

Input circuit

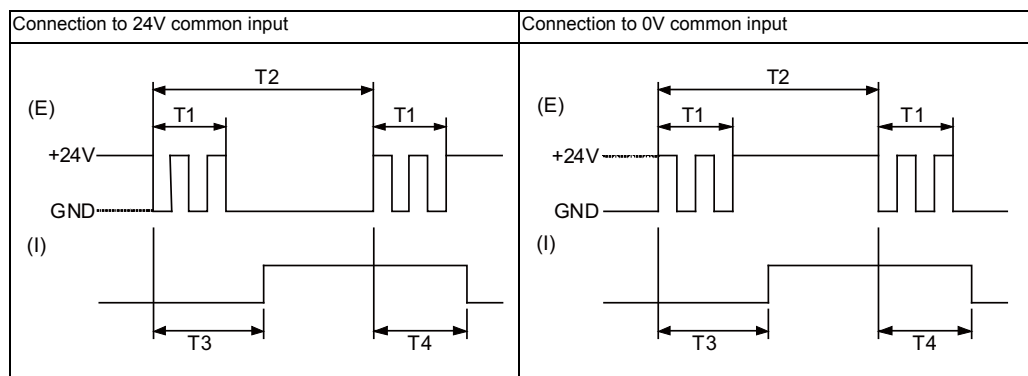


Input conditions

The input signals must be used within the following condition ranges.

		24V common	0V common
1	Input voltage at external contact ON	6V or less	18V or more, 25.2V or less
2	Input current at external contact ON	9mA or more	
3	Input voltage at external contact OFF	20V or more, 25.2V or less	4V or less
4	Input current at external contact OFF	2mA or less	
5	Input resistance	Approx. 2.2kΩ	
6	Tolerable chattering time (T1)	3ms	
7	Input signal holding time (T2)	40ms or more (Note)	
8	Input circuit operation delay time (T3 and T4)	3 to 16ms	
9	Machine side contact capacity	30V or more, 16mA or more	

(Note) Input signal holding time: The guide is 40ms or more. The input signal will not be recognized unless it is held for the ladder processing cycle time or longer.



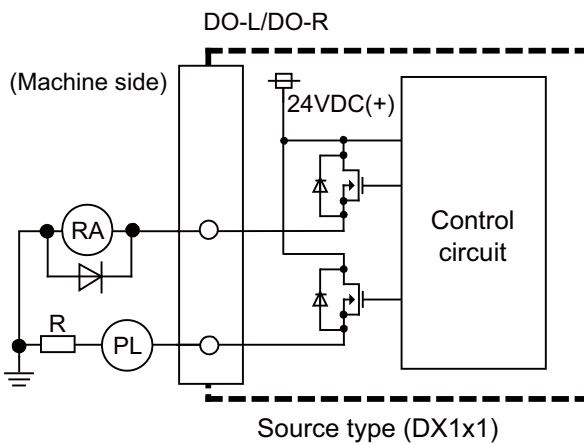
(E) : External signal, (I) : Internal signal

(3) Machine output (DO-L)

(4) Machine output (DO-R)

Use within the specification ranges shown below.

Output circuit



Output conditions

Insulation method	Non-insulation
Rated load voltage	24VDC
Max. output current	60mA/point
Output delay time	40μs

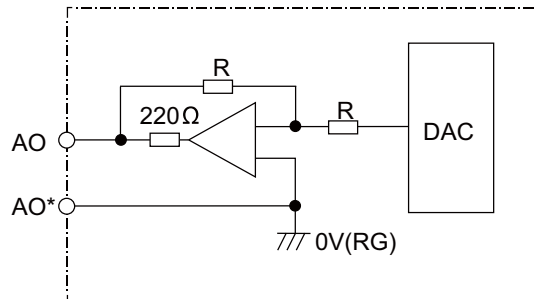
(Note 1) When using an inductive load such as a relay, always connect a diode (voltage resistance 100V or more, 100mA or more) in parallel to the load.

(Note 2) When using a capacitive load such as a lamp, always connect a protective resistor (R=150Ω) serially to the load to suppress rush currents. (Make sure that the current is less than the above tolerable current including the momentary current.)

[Outline of analog signal output circuit]

The analog signal output circuit can be used only for FCUA-DX121.

Output circuit



Output conditions

Output voltage	0V to ±10V (±5%)
Resolution	12bit (±10V × n/4096) (Note)
Load conditions	10kΩ load resistance
Output impedance	220Ω

(Note) $n=(2^0 \text{ to } 2^{11})$

⚠ CAUTION

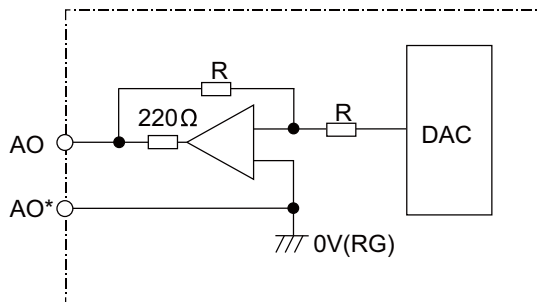
1. When using an inductive load such as a relay, always connect a diode in parallel to the load.
2. When using a capacitive load such as a lamp, always connect a protective resistor serially to the load to suppress rush currents.

(5) Analog signal input/output (AIO)

(a) Outline of analog signal output circuit

The analog signal output circuit can be used only for FCUA-DX141.

Output circuit



Output conditions

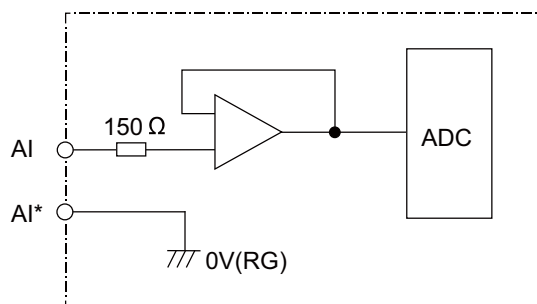
Output voltage	0V to ±10V (±5%)
Resolution	12bit (±10V × n/4096) (Note)
Load conditions	10kΩ load resistance
Output impedance	220Ω

(Note) $n=(2^0 \text{ to } 2^{11})$

(b) Outline of analog signal input circuit

The analog signal input circuit can be used only for FCUA-DX141.

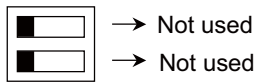
Input circuit



Input conditions

Max. input rating	±15V
Resolution	$ -10 \text{ to } +10\text{V} / 4096 = 4.88\text{mV}$
Precision	Within ±25mV
AD input sampling time	14.2ms(AI0) / 42.6ms(AI1 to 3)

(6) Transfer speed changeover switch (DS)

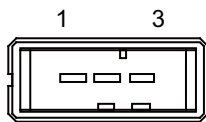


(7) Station No. changeover switch (CS)



(8) Remote I/O unit I/F #1 (RIO1)

(9) Remote I/O unit I/F #2 (RIO2)



1	I/O	TXRX
2	I/O	TXRX*
3		0V

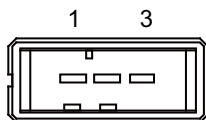
<Cable side connector type>

Connector: 1-178288-3

Contact: 1-175218-2 x3

Recommended manufacturer: Tyco Electronics

(10) 24VDC input (DCIN)



1	I	+24V
2		0V
3		FG

<Cable side connector type>

Connector: 2-178288-3

Contact: 1-175218-5 x3

Recommended manufacturer: Tyco Electronics

(11) LED

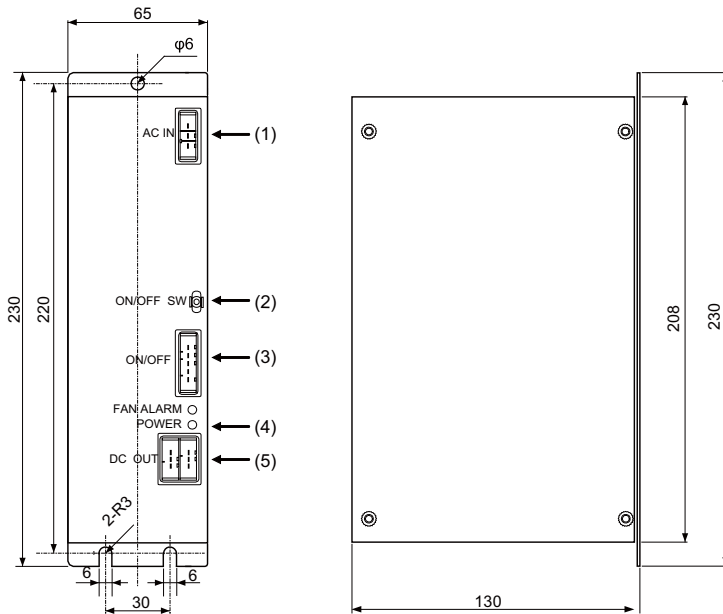
Name	Function	At fault	Conditions
POWER Green (UP)	Internal output voltage(5VDC) check	Not lit	(1) Failure of 24VDC input (2) Fuse is disconnected (3) Failure of internal voltage output in I/O unit
ALM Red (DOWN)	RIO communication status check	Lit	(1) Partial disconnection between control unit and remote I/O unit (2) Miswiring of connected remote I/O unit

5.6 External Power Supply Unit

Item	PD25
Input power voltage	200 to 230VAC +10%-15% 50/60Hz±1Hz
Output current	3A
Dimension	130mm × 65mm × 230mm
Mass	1.5kg
Output holding time	300ms

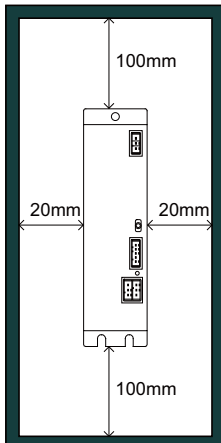
- (Note 1) PD25 will not be turned ON by the ON/OFF switch immediately after the power OFF. Wait at least 2 seconds, and then turn the power ON.
- (Note 2) The power supply configuration for NC devices depends on the machine. Appropriate circuit protector must be selected by machine builder according to the load of the machine.
- (Note 3) PD25 does not comply with 100VAC.

Dimension and names of parts
[PD25]

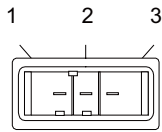


<Mounting direction and clearance>

Mount the external power supply unit vertically and so that it is visible from the front. Provide space for heat dissipation and ventilation.



(1) AC power input (ACIN)



1	I	ACIN N
2	I	ACIN H
3		FG

<Cable side connector type>

Connector: 2-178128-3

Contact: 1-175218-5 × 3

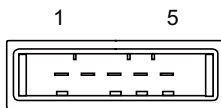
Recommended manufacturer: Tyco Electronics

(2) ON/OFF switch (ON/OFF SW)

Switch ON (upward): 24VDC output

Switch OFF (downward): 24VDC output OFF

(3) ON/OFF input (ON/OFF)



1		NC
2	I	P-OFF
3		0V
4		0V
5	I	P-ON

<Cable side connector type>

Connector: 1-178288-5

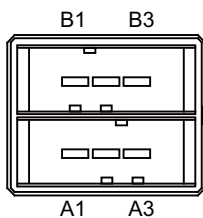
Contact: 1-175218-5 × 4

Recommended manufacturer: Tyco Electronics

(4) LED (POWER)

Green light at +24V output

(5) 24VDC output (DCOUT)



1A	O	ACFAIL	1B	O	+24V
2A		0V	2B		0V
3A		NC	3B		FG

<Cable side connector type>

Connector: 3-178127-6

Contact: 1-175218-5 (for AWG16) × 3, 1-175217-5 (for AWG22) × 2

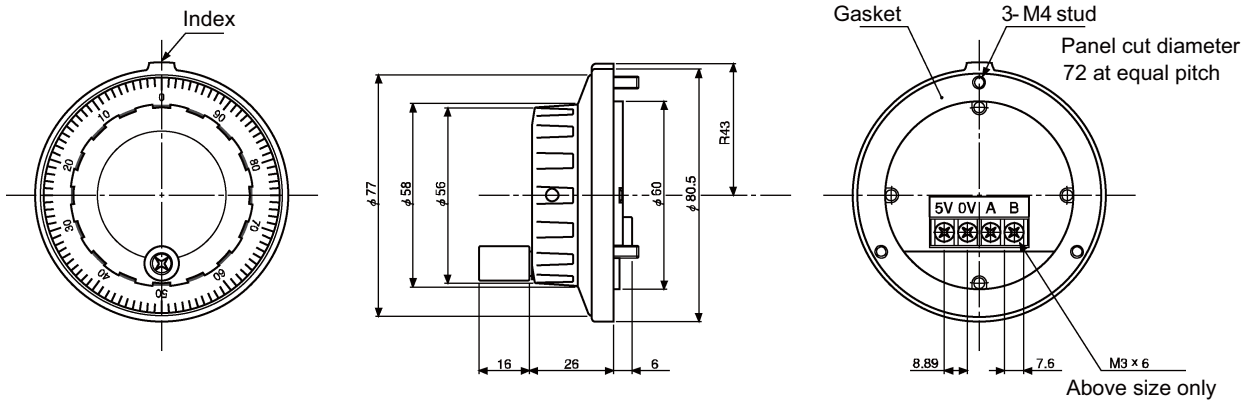
Recommended manufacturer: Tyco Electronics

5.7 Manual Pulse Generator

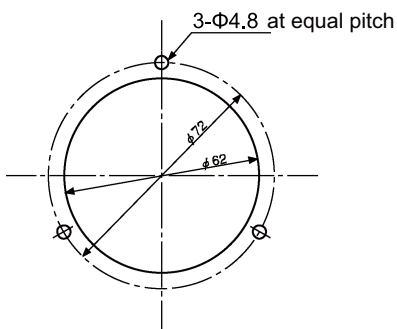
[UFO-01-2Z9]

5V Manual Pulse Generator (100 pulse/rev)

<Outline dimension drawing>



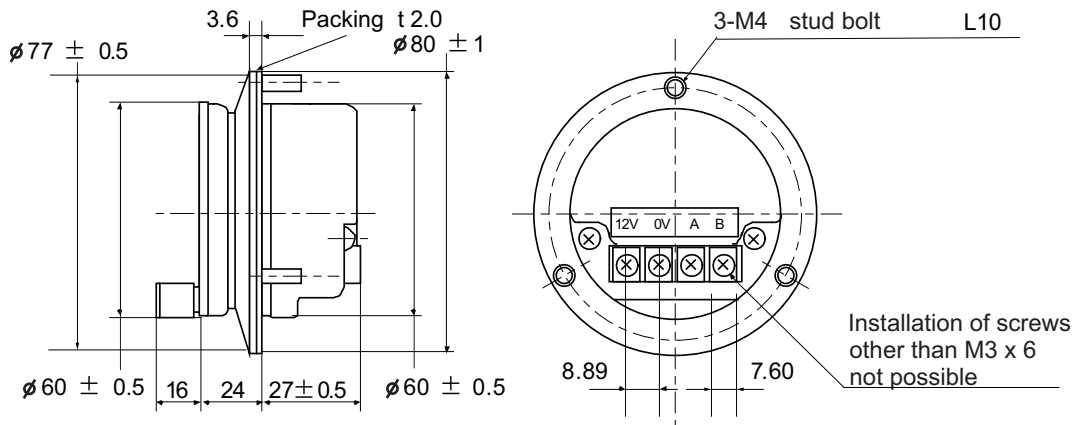
<Panel cut drawing>



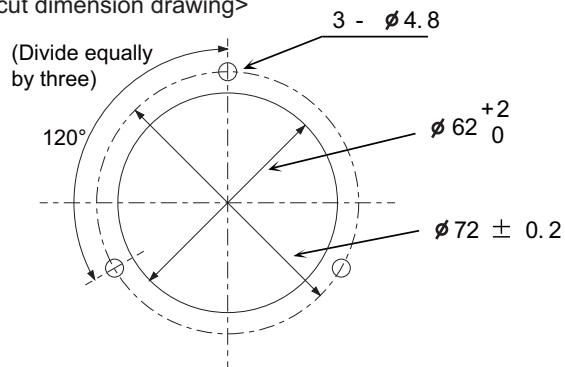
Produced by NIDEC NEMICON CORPORATION

[HD60C]

12V Manual Pulse Generator (25 pulse/rev)



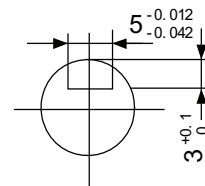
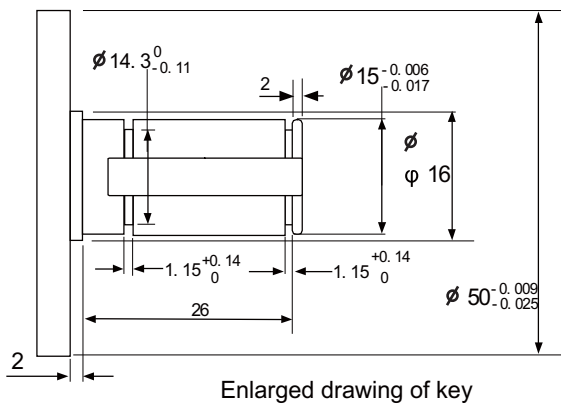
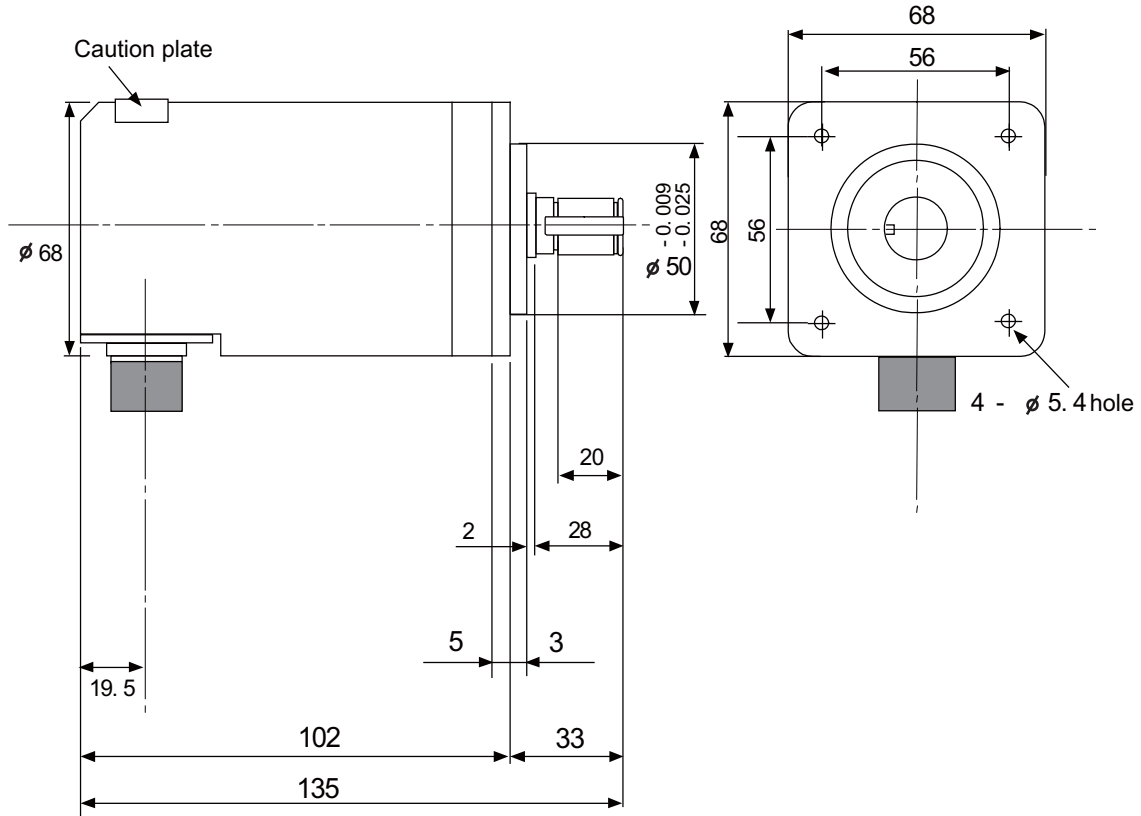
<Panel cut dimension drawing>



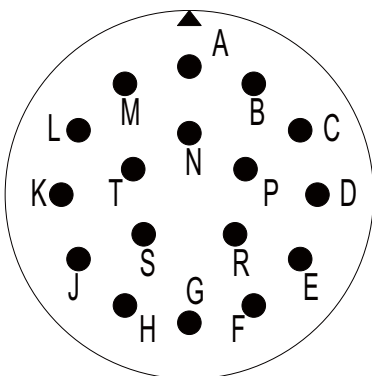
5.8 Synchronous Feed Encoder

Dimension and names of parts

[OSE-1024-3-15-68]



Cross section BB
Valid depth of key groove is 21mm



Connector pin assignment

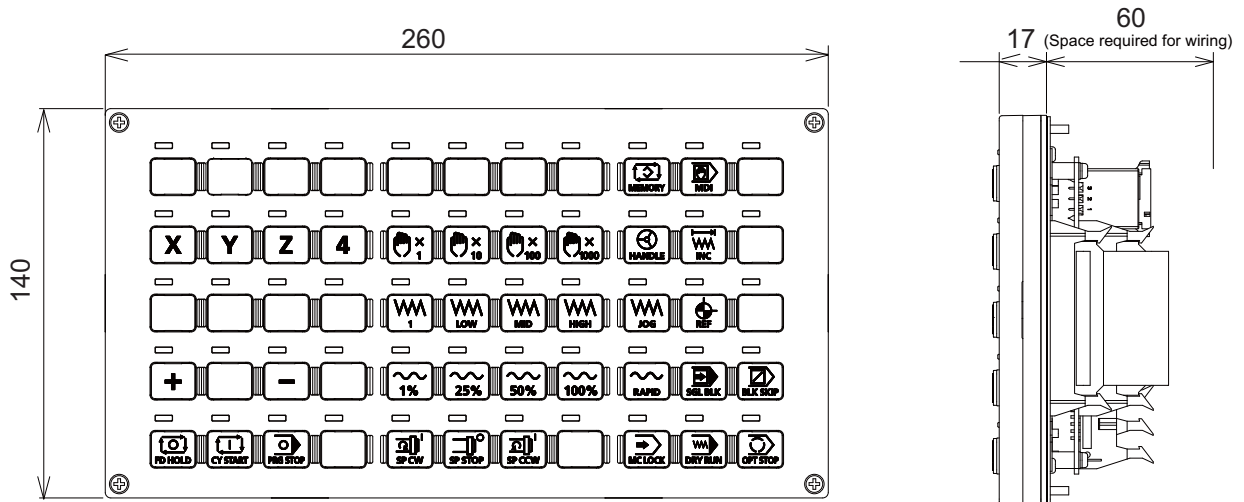
Pin	Function	Pin	Function
A	A phase	K	0V
B	Z phase	L	-
C	B phase	M	-
D	-	N	A phase
E	Case grounding	P	Z phase
F	-	R	B phase
G	-	S	-
H	+5V	T	-
J	-		

5.9 MITSUBISHI CNC Machine Operation Panel

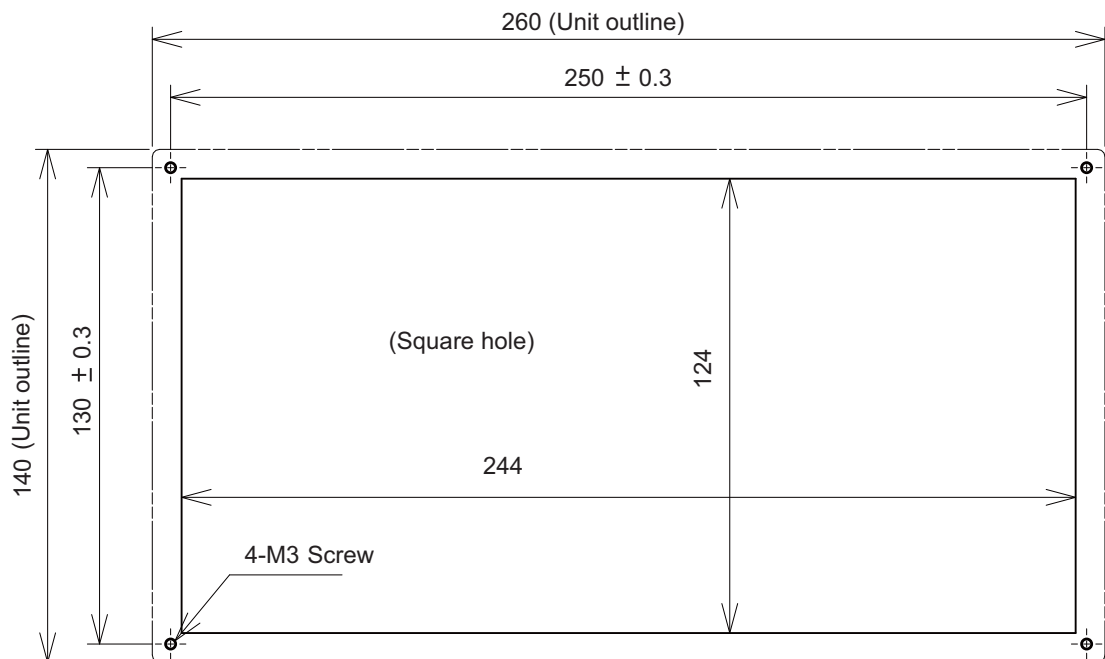
5.9.1 MITSUBISHI CNC Machine Operation Panel A

Outline dimension

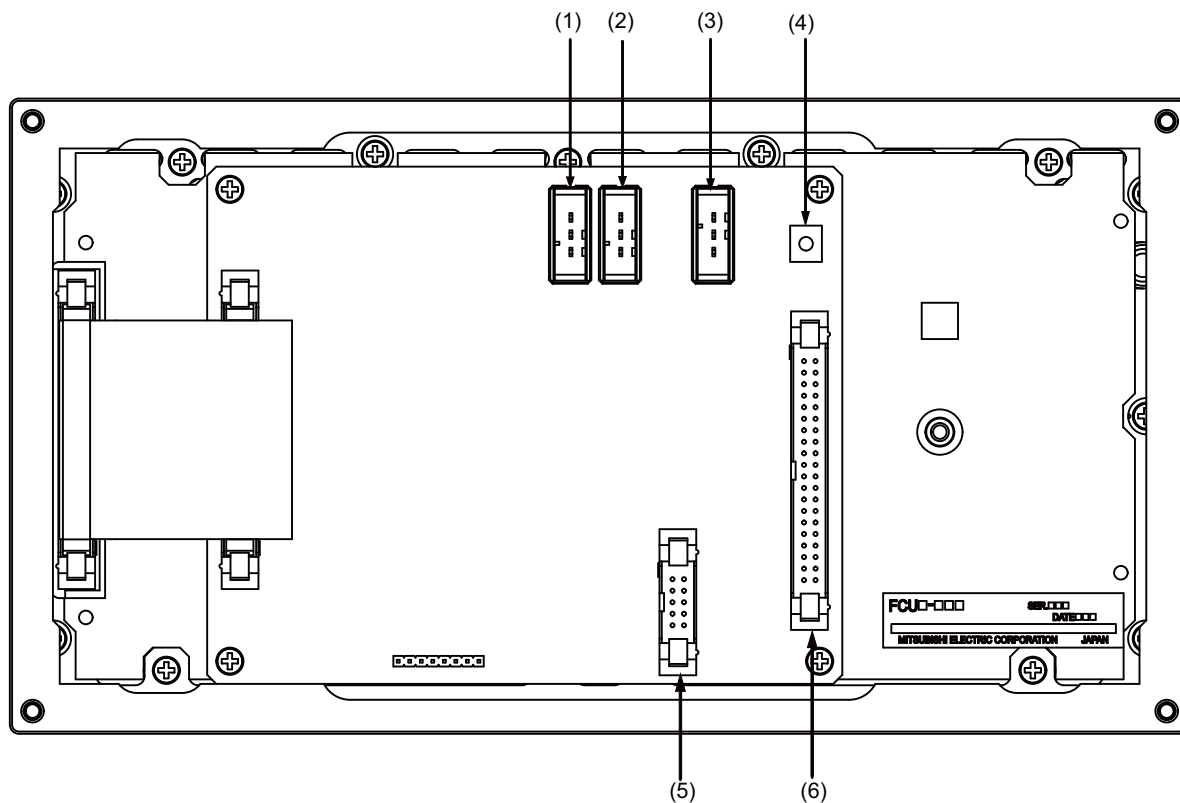
[FCU7-KB921 (MITSUBISHI CNC machine operation panel A)]



<Panel cut dimension drawing>

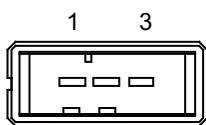


Connector



No.	Connector name	Function	No.	Connector name	Function
(1)	DCIN	24VDC input	(4)	FG	FG terminal
(2)	DCOUT	24VDC output	(5)	EXT	Relay wiring
(3)	RIOIN	Remote I/O	(6)	SUBP	Machine operation panel B switch

(1)(2) 24VDC input/output (DCIN/DCOUT)



1	I/O	+24V
2	-	0V
3	-	FG

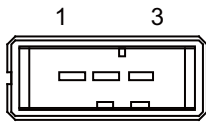
<Cable side connector type>

Connector: 2-178288-3

Contact: 1-175218-5 x3

Recommended manufacturer: Tyco Electronics

(3) Remote I/O connected connector (RIOIN)



1	I/O	TXRX
2	I/O	TXRX*
3	-	0V

<Cable side connector type>

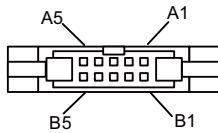
Connector: 1-178288-3

Contact: 1-175218-2 x3

Recommended manufacturer: Tyco Electronics

(4) FG terminal (FG)

(5) Relay wiring (EXT)



B05	-		A05	-	
B04	O	Relay EMGCOM	A04	O	Relay EMG
B03	O	Relay 6	A03	O	Relay 5
B02	O	Relay 4	A02	O	Relay 3
B01	O	Relay 2	A01	O	Relay 1

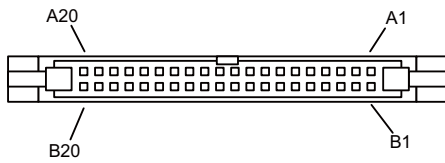
<Cable side connector type>

Connector: HIF3BA-10D-2.54C

Contact: HIF3-2428SC

Recommended manufacturer: Hirose Electric

(6) Machine operation panel B switch (SUBP)



B20	I	Relay EMGCOM	A20	I	Relay EMG
B19	I	Relay 6	A19	I	Relay 5
B18	I	Relay 4	A18	I	Relay 3
B17	I	Relay 2	A17	I	Relay 1
B16	-	0V	A16	I	Spare SW(X75)
B15	-	0V	A15	I	Spare SW(X74)
B14	-	0V	A14	I	Spare SW(X73)
B13	-	0V	A13	I	Selector SW(X72)
B12	-		A12	I	G(X71)
B11	I	F(X6D)	A11	I	E(X6F)
B10	I	D(0V)	A10	I	C(X70)
B09	I	B(X6E)	A09	I	A(X6C)
B08	-		A08	I	G(X6B)
B07	I	F(X67)	A07	I	E(X66)
B06	I	D(0V)	A06	I	C(X6A)
B05	I	B(X68)	A05	I	A(X66)
B04	-		A04	I	G(X65)
B03	I	F(X61)	A03	I	E(X63)
B02	I	D(0V)	A02	I	C(X64)
B01	I	B(X62)	A01	I	A(X60)

<Cable side connector type>

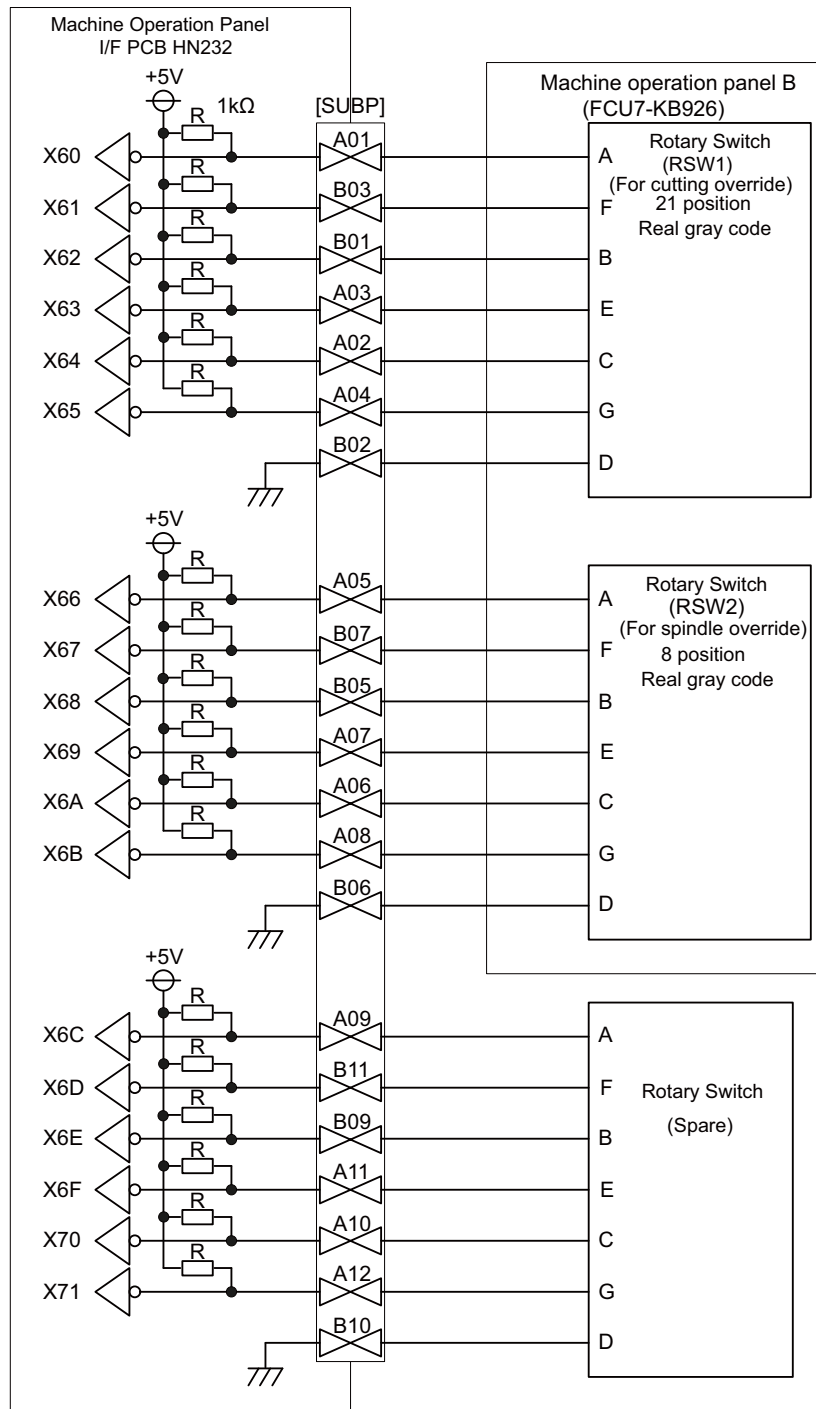
Connector: HIF3BA-40D-2.54C

Contact: HIF3-2428SC

Recommended manufacturer: Hirose Electric

(a) Wiring of machine operation panel B

<Wiring of rotary switch (SUBP)>



(Note1) Use the special lead wire with a connector for wiring of rotary switch.

Lead wire with a connector: ACX011-705 (7 terminals, 0.5m) Fuji Electric

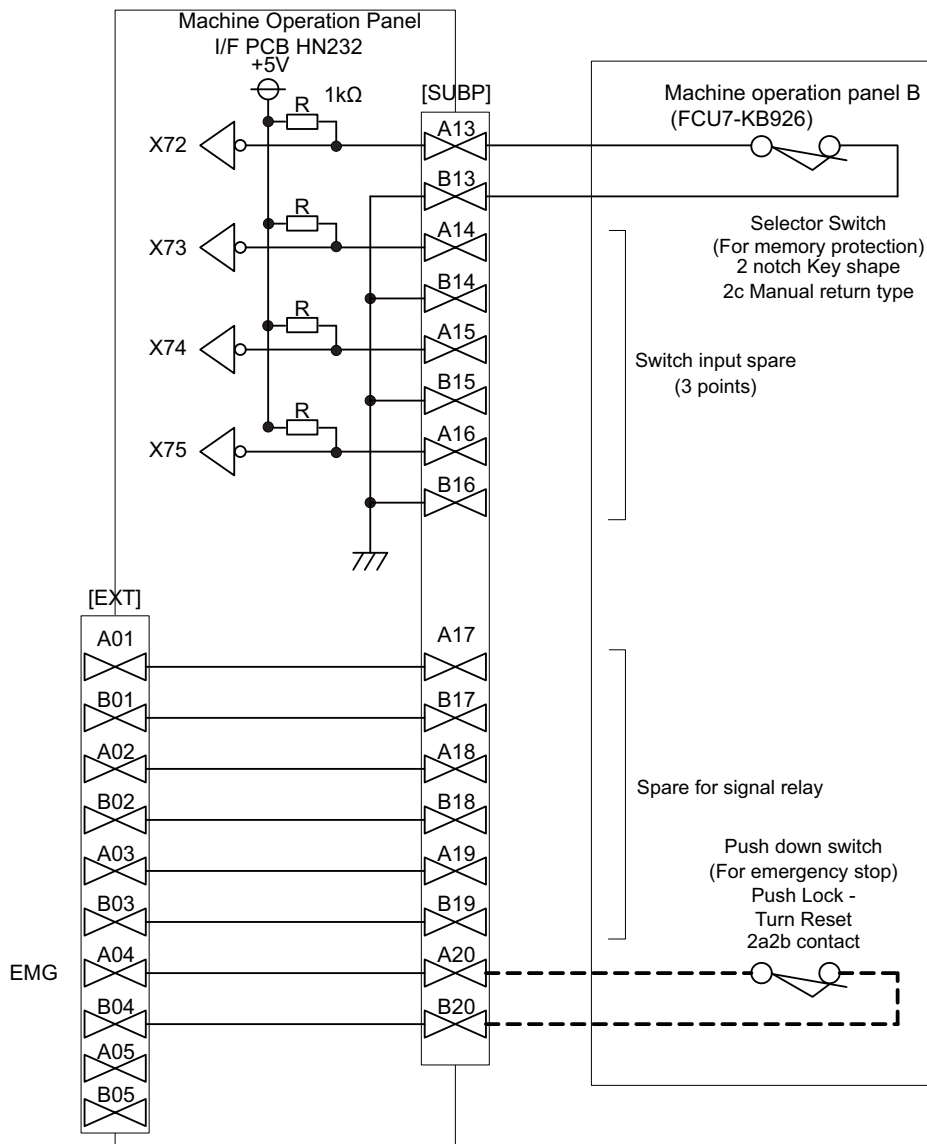
(Note2) A09 to 12 and B09 to 11 of SUBP are spares, so they do not need wiring.

(Note3) Select a rotary switch which guarantees 5V/1mA.

(Note4) Length of cables must be 0.5m or shorter.

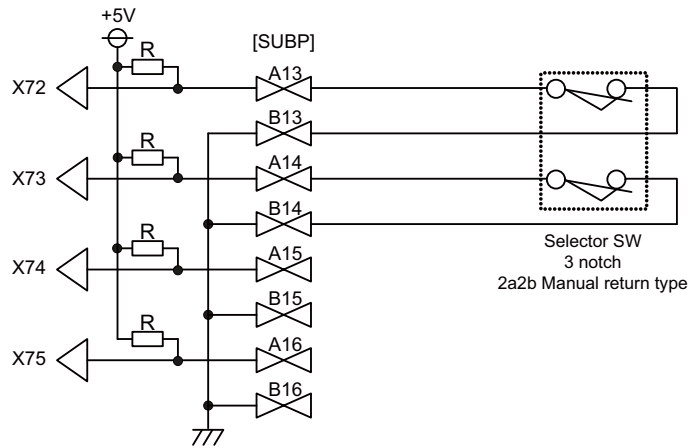
(Note5) The numbers in the above diagram do not indicate the actual device numbers.

<Wiring of other switches (SUBP)>

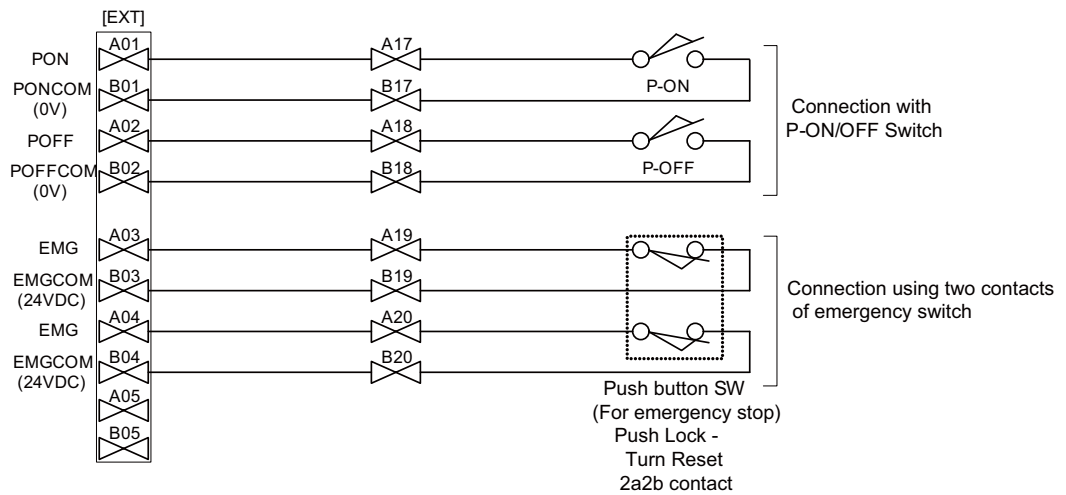


- (Note1) Wirings for selector switches must be directly soldered to terminals or use tab terminals (110 series).
- (Note2) When wiring the emergency stop switch, the crimp terminal must be fastened with thread.
- (Note3) For NCs whose control unit and display unit are integrated, wire the emergency stop switch directly to the EMG connector of the control unit. (Use F120 cable)
And for NCs whose control unit and display are separated, relay it by the machine operation panel I/F PCB HN232.
- (Note4) Select a rotary switch which guarantees 5V/1mA.
- (Note5) Length of cables must be 0.5m or shorter.
- (Note6) The numbers in the above diagram do not indicate the actual device numbers.

<Connection when the selector SW has 3 notches>



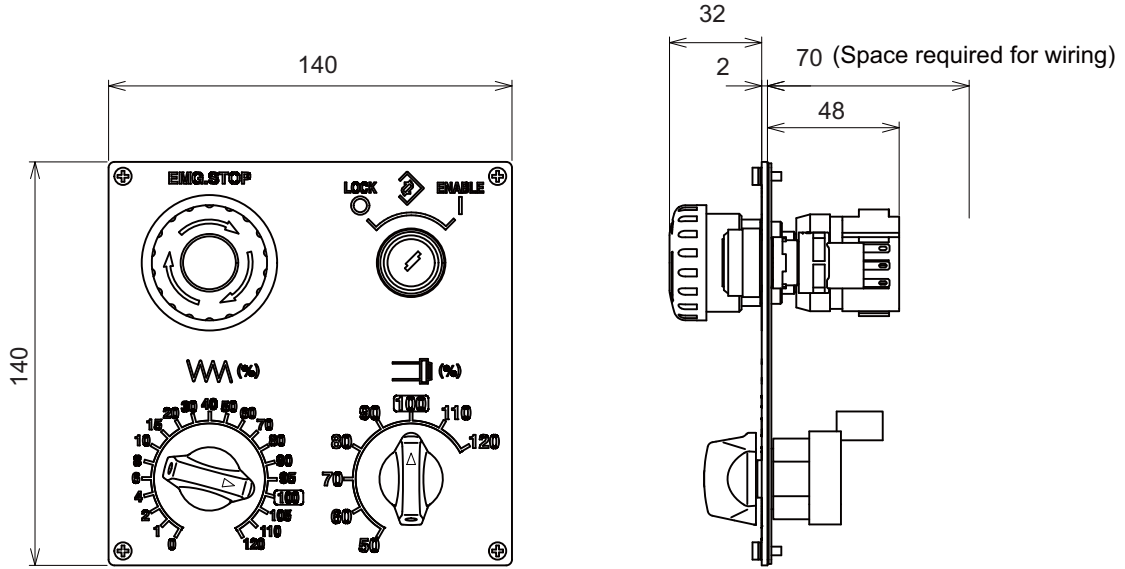
<Connection when relaying wiring to control panel>



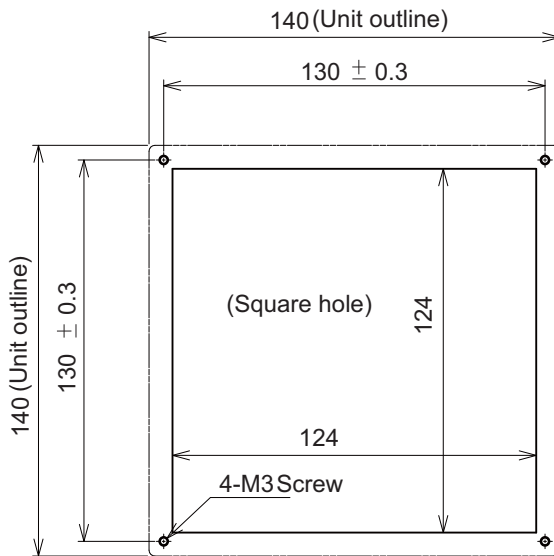
5.9.2 MITSUBISHI CNC Machine Operation Panel B

Outline dimension

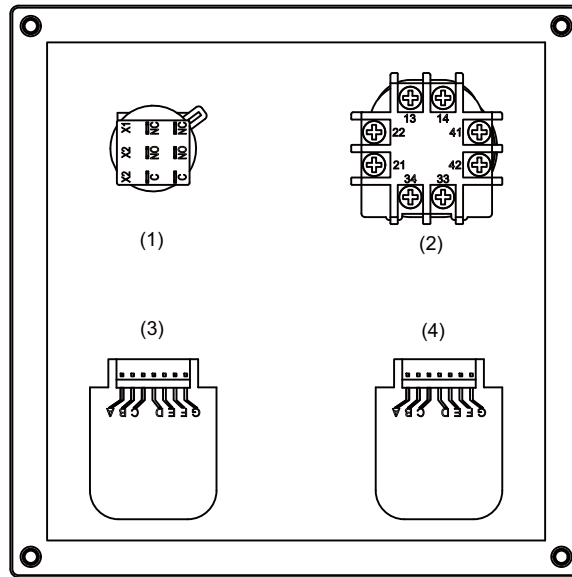
[FCU7-KB926 (MITSUBISHI CNC machine operation panel B)]



<Panel cut drawing>

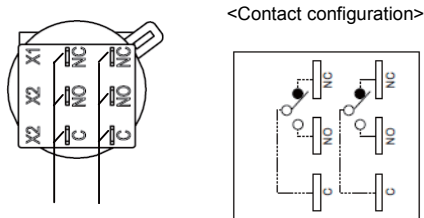


Connector



No.	Switch	No.	Switch
(1)	Memory protection switch	(3)	Spindle override (RSW2)
(2)	Emergency stop button	(4)	Cutting override (RSW1)

(1) Memory protection switch



Contact 2 Contact 1

C	-	Contact 1(Common)
NO	O	Contact 1(A contact)
NC	O	Contact 1(B contact)
C	-	Contact 2(Common)
NO	O	Contact 2(A contact)
NC	O	Contact 2(B contact)

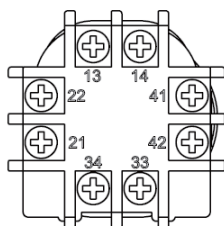
<Switch type>

Switch: HA1K-2C2B

Recommended manufacturer: IDEC

Connection: Terminal (#110 type)

(2) Emergency stop button



13	O	Contact 1(A contact)
14	O	Contact 1(A contact)
21	O	Contact 2(B contact)
22	O	Contact 2(B contact)
33	O	Contact 3(A contact)
34	O	Contact 3(A contact)
41	O	Contact 4(B contact)
42	O	Contact 4(B contact)

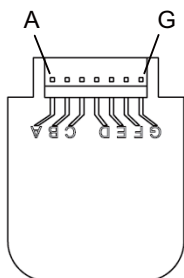
<Switch type>

Switch: XA1E-BV422MR

Recommended manufacturer: IDEC

Connection: Thread terminal (M3)

(3) Spindle override (RSW2)



A	O	Output signal 1
B	O	Output signal 3
C	O	Output signal 5
D	-	Common terminal
E	O	Output signal 4
F	O	Output signal 2
G	-	-

<Switch type>

Switch: AC09-GX0/7L3B02

Recommended manufacturer: Fuji Electric

Connection: Connector

<Switch side connector type>

Connector: IL-7P-S3EN2

Recommended manufacturer: Japan Aviation Electronics

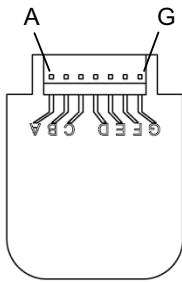
<Cable side connector type>

Connector: IL-7S-S3L-(N)

Contact: L-C2-10000

Recommended manufacturer: Japan Aviation Electronics

(4) Cutting override (RSW1)



A	O	Output signal 1
B	O	Output signal 3
C	O	Output signal 5
D	-	Common terminal
E	O	Output signal 4
F	O	Output signal 2
G	O	Output signal 6

<Switch type>

Switch: AC09-GY0/20L3B02

Recommended manufacturer: Fuji Electric

Connection: Connector

<Switch side connector type>

Connector: IL-7P-S3EN2

Recommended manufacturer: Japan Aviation Electronics

<Cable side connector type>

Connector: IL-7S-S3L-(N)

Contact: IL-C2-10000

Recommended manufacturer: Japan Aviation Electronics

5.10 Exclusive CF Cards for MITSUBISHI CNC

Item	FCU7-CF256M	FCU7-CF002G
Capacity	256MB	2GB
Operation-guaranteed temperature	-40 °C to +85 °C	
NAND Flash	SLC (Note)	

(Note) SLC stands for Single Level Cell, and it stores one bit data in each memory cell. This provides longer life span and high product reliability in comparison with MLC (Multi Level Cell), which is commonly applied to CF cards.

5.10.1 Precautions for Use of Commercially Available CF Cards

Mitsubishi is unable to guarantee the machine operation when a commercially available CF card or SD memory card (SD-CF adapter is required) is used. In that case, performance check must be made carefully by machine tool builder.

- (1) Commercially available CF cards may not be compatible with MITSUBISHI units or suitable FA environment for temperature- or noise-wise. In case of using it, careful performance check must be required by the machine tool builder.
- (2) When inserting/removing a commercially available CF card, turn the MITSUBISHI device's power OFF to avoid any troubles. If a card must be inserted and removed while the power is ON, make sure to take sufficient time (approx. ten seconds or more) between the insertion and removal.
- (3) Do not pull out the card or turn OFF the power during access to the CF card. Failure to observe this could cause the memory contents to be erased. In case of emergency, always perform backups by having your important data duplicated, etc. as MITSUBISHI will not guarantee the broken or lost data.

II Functional Specifications



Control Axes

1.1 Control Axes

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

The NC axis is an axis that can be manually operated, or automatically operated with the machining program. X, Y, Z, U, V, W, A, B and C axis can be used.

The PLC axis is an axis that can be controlled from the PLC ladder.

1.1.1 Number of Basic Control Axes (NC Axes)

M system	○ 3
L system	○ 2

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

M system	6
L system	6

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 optical servo channel (OPT).

Refer to the Connection Manual for details.

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

M system	3
L system	3

1.1.2.2 Max. Number of Spindles

M system	1
L system	2

Includes analog spindles.

1.1.2.3 Max. Number of PLC Axes

M system	2
L system	3

1.1.4 Max. Number of PLC Indexing Axes

M system	1
L system	1

PLC axis indexing function: This function 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

M system	3
L system	3

1.2 Control Part System

1.2.2 Max. Number of Part Systems

M system	<input type="radio"/> 1
L system	<input type="radio"/> 1

1.3 Control Axes and Operation Modes

1.3.1 Tape (RS-232C Input) Mode

M system	<input type="radio"/>
L system	<input type="radio"/>

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

M system	<input type="radio"/>
L system	<input type="radio"/>

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

1.3.3 MDI Mode

M system	<input type="radio"/>
L system	<input type="radio"/>

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.

1.3.5 Front IC Card Mode

M system	<input type="radio"/>
L system	<input type="radio"/>

The machining program stored in a compact flash card can be operated by installing a compact flash card on the front of control unit.

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

Also, when "M98 Pp ,Dd;" ("d" for designating a unit) is commanded in the main program, the machining program in a compact flash card can be called and operated as a sub program.

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

2

Input Command

2.1 Data Increment

2.1.1 Least Command Increment

The data increment handled in the controller includes the input setting increment and command increment. Each type is set with parameters.

- (1) The input setting increment indicates the increment handled in the internal processing of the controller. The counter and tool compensation data, etc., input from the screen is handled with this increment. This increment is also applied to a 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 = °)
1µm (B)	0.001	0.001	0.0001	0.001
0.1µm (C)	0.0001	0.0001	0.00001	0.0001

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

- (2) The command increment indicates the command increment of the movement command in the machining program. This can be set per axis.

Command increment (parameter)	Metric unit system		Inch unit system	
	Linear axis (Unit = mm)	Rotary axis (Unit = °)	Linear axis (Unit = inch)	Rotary axis (Unit = °)
0	Accordance with #1003 iunit.			
1	0.0001	0.0001	0.00001	0.0001
10	0.001	0.001	0.0001	0.001
100	0.01	0.01	0.001	0.01
1000	0.1	0.1	0.01	0.1
10000	1.0	1.0	0.1	1.0

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

2.1.1.1 Least Command Increment 1 µ m (Input Setting Increment 1 µ m)

M system	<input type="radio"/>
L system	<input type="radio"/>

It is possible to command 0.001mm for the linear axis and 0.001° for the rotation axis.

2.1.1.2 Least Command Increment 0.1 µ m (Input Setting Increment 0.1 µ m)

M system	<input type="radio"/>
L system	<input type="radio"/>

It is possible to command 0.0001mm for the linear axis and 0.0001° for the rotation axis.

2.1.2 Least Control Increment

The least control increment determines the CNC's internal operation accuracy.

2.1.2.1 Least Control Increment 0.01 μ m (10nm)

M system	<input type="radio"/>
L system	<input type="radio"/>

2.1.2.2 Least Control Increment 0.001 μ m (1nm)

M system	<input type="radio"/>
L system	<input type="radio"/>

2.2 Unit System

2.2.1 Inch/Metric Changeover

M system	<input type="radio"/>
L system	<input type="radio"/>

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 option is required when the unit is switched with the machining program command.

Unit system	Length data	Meaning
Metric unit system	1.0	1.0mm
Inch unit system	1.0	1.0inch

(Note) For the angle data, 1.0 means 1 degree ($^{\circ}$) regardless of the unit system.

Parameter	Data					
	Machining program		Screen data (Compensation amount, user parameter, counter, etc.) / Feedrate of handle, etc.	Machine parameter / PLC interface machine position, etc.		
A	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			
B	0	Not affected		Not affected	Metric unit system	
	1	Not affected		Not affected	Inch unit system	

(Note 1) The parameter changeover is valid after the power is turned ON again.

(Note 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).

(Note 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

M system	○
L system	—

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 μ m, to run a machining program, which has been created with a 10 μ m 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_);

(Note 2) This function cannot be used for the compensation amount of the tool offset input.

(Note 3) This function can be used when decimal point type I is valid, but cannot be used when decimal point type II is valid.

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)

M system	—
L system	○

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

M system	—
L system	○

2.3.1.4 Format 1 for Machining Center

M system	○
L system	—

2.4 Command Value

2.4.1 Decimal Point Input I, II

M system	○
L system	○

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

(1) 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.

(2) 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	X100mm, Y-200.5mm	<-
G1 X100 F20.	X100μm, F20mm/min	X100mm, F20mm/min
G1 Y200 F100 (*1)	Y200μm, F100mm/min	Y200mm, F100mm/min
G4 X1.5	Dwell 1.5s	<-
G4 X2	Dwell 2ms	Dwell 2s

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

2.4.2 Absolute/Incremental Command

M system	<input type="radio"/>
L system	<input type="radio"/>

(1) 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.

G90 ... Absolute command (absolute command)

G91 ... Incremental command (incremental command)

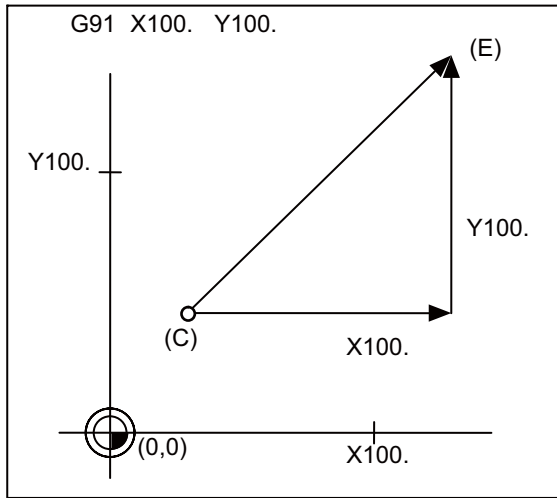
These G codes can be commanded multiple times in one block.

Example

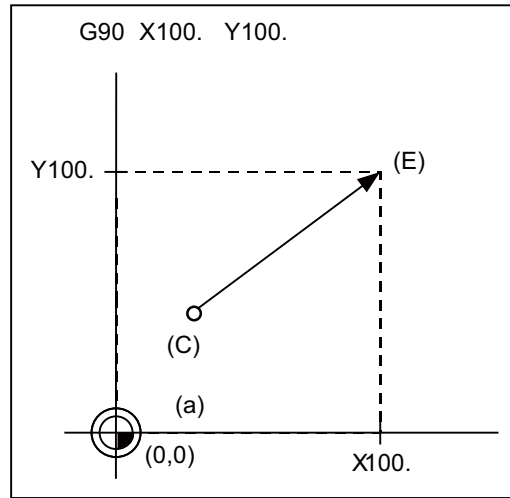
G90 X100.	G91 Y200.	G90 Z300.	;
Absolute position	Incremental position	Absolute position	

(Note 1) As with the memory command, if there is no G90/G91 designation in the MDI command, the previously executed modal will be followed.

(Incremental command)



(Absolute command)



(a) Program coordinate
 (C) Current position
 (E) End point

(2) 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.

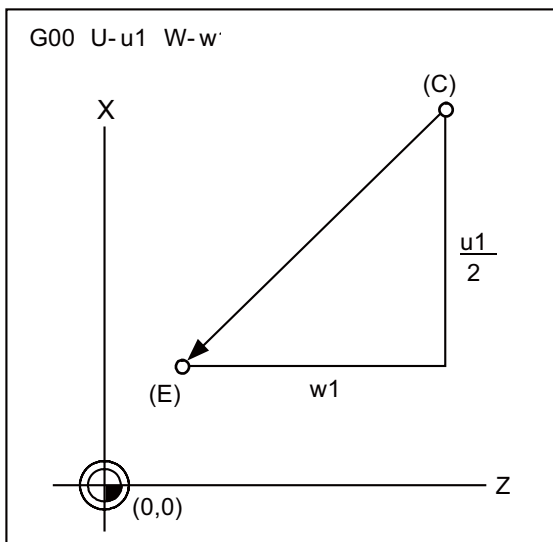
Absolute command (absolute command) ... X, Z

Incremental command (incremental command) ... U, W

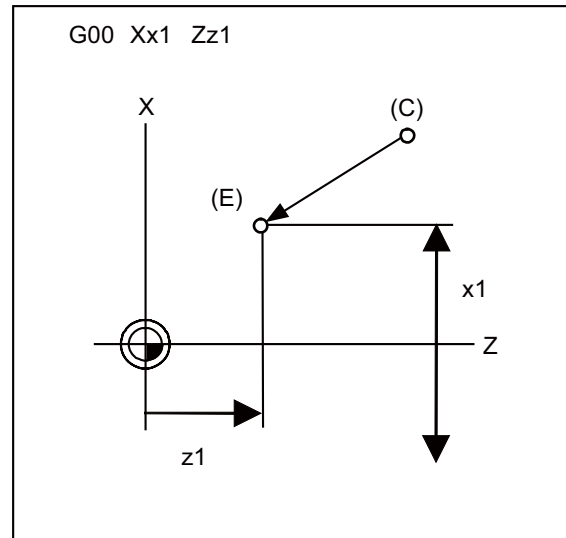
Example

G00 X100.	W200.	;
Absolute position	Incremental position	

(Incremental command)



(Absolute command)



(a) Program coordinate

(C) Current position

(E) End point

The above drawing shows the case for the diameter command.

(Note) 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) may be selected.

2.4.3 Diameter/Radius Designation

M system	—
L system	○

For axis command value, the radius designation or diameter designation can be changed over with parameters.

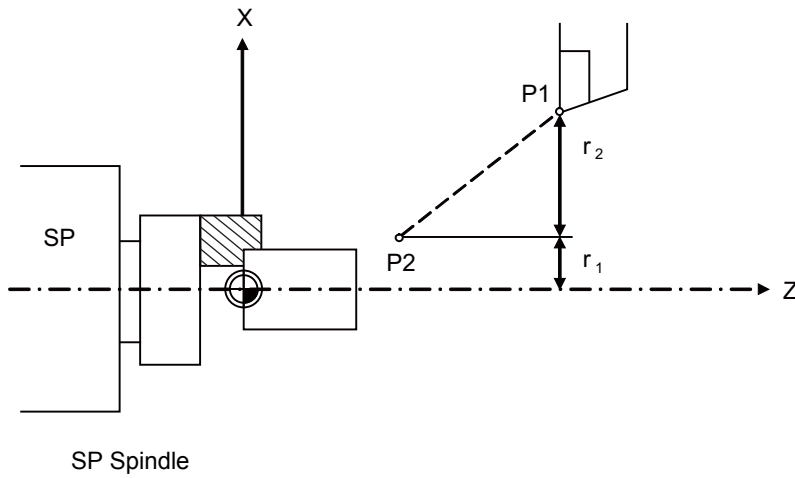
When the diameter designation is selected, the scale of the length of the selected axis is doubled.

(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



Radius and diameter commands

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

Positioning/Interpolation

3.1 Positioning

3.1.1 Positioning

M system	○
L system	○

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 rapid traverse feed rate of the NC, refer to the section entitled "Rapid Traverse Rate".

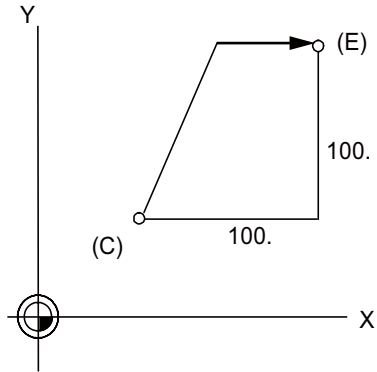
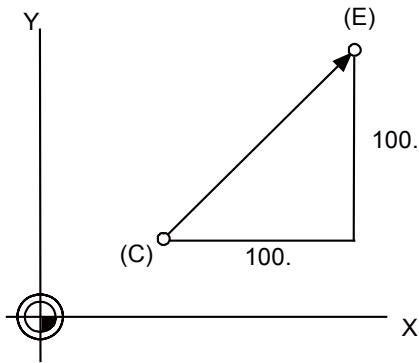
Since the actual rapid traverse feed rate depends on the machine, refer to the specifications of the machine concerned.

- (1) The rapid traverse feed rate can be set for each axis with parameters.
- (2) The number of axes which can be commanded simultaneously depends on the specifications (number of simultaneously controlled axes). The axes can be used in any combination within this range.
- (3) 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)

(Example) Linear type (Moves linearly to the end point.)
 G00 G91 X100. Y100. ;

(Example) Non-linear type (Each axis moves at each parameter speed.)
 G00 G91 X100. Y100. ;



(C) Current position
 (E) End point

- (4) The tool is always accelerated at the start of the program command block and decelerated at the end of the block.

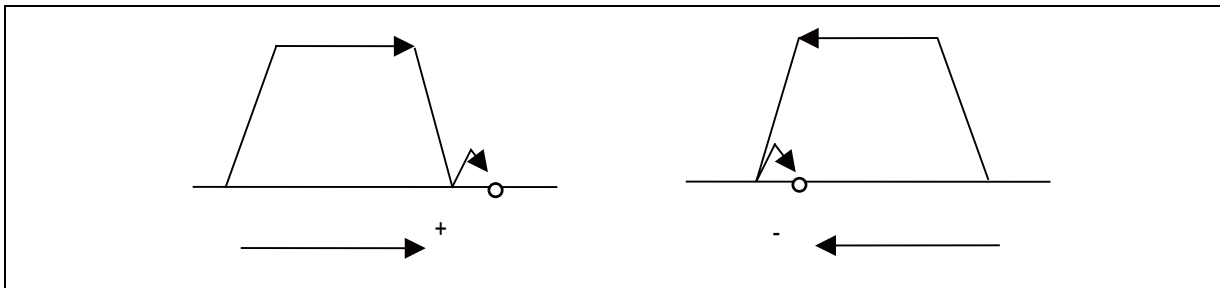
3.1.2 Unidirectional Positioning

M system	○
L system	—

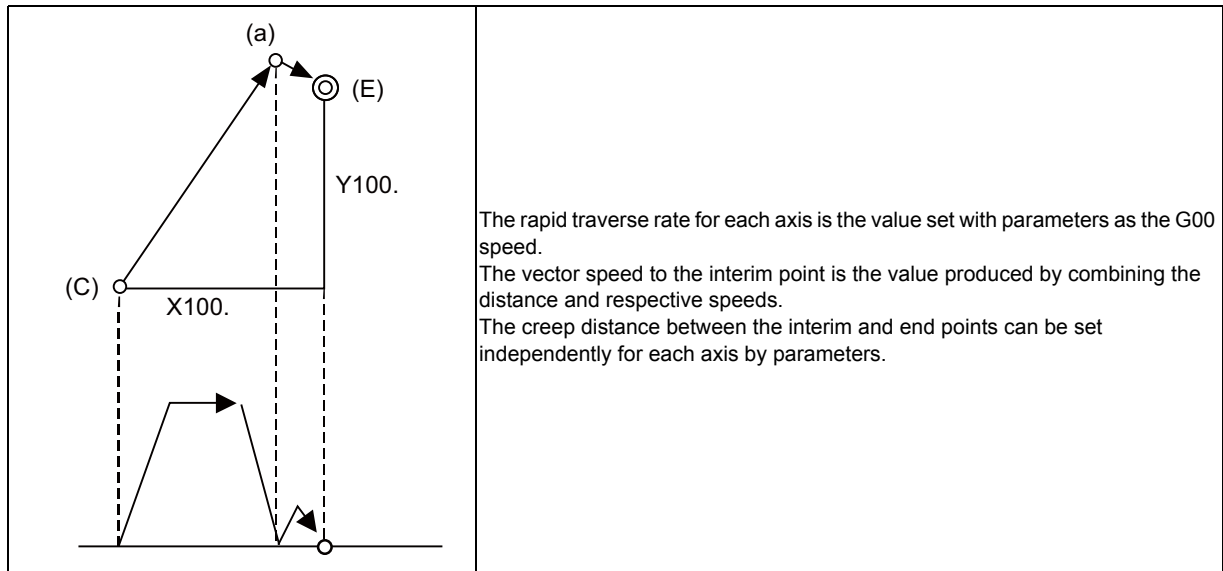
The G60 command always moves the tool to the final position in a direction determined with parameters. The tool can be positioned without backlash.

G60 Xx1 Yy1 Zz1 ;(Also possible for additional axes A, B, C, U, V, W simultaneously)
Xx1,Yy1,Zz1 :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 rapid traverse feed rate of the NC, refer to the section entitled "Rapid Traverse Rate". 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.)



(Example)
G60 G91 X100. Y100. ;



- (Note 1) The processing of the above pattern will be followed even for the machine lock and Z-axis command cancel.
- (Note 2) On the creep distance, the tool is moved with rapid traverse.
- (Note 3) G60 is valid even for positioning in drilling in the fixed cycle.
- (Note 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

M system	○
L system	○

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 :Feed rate data

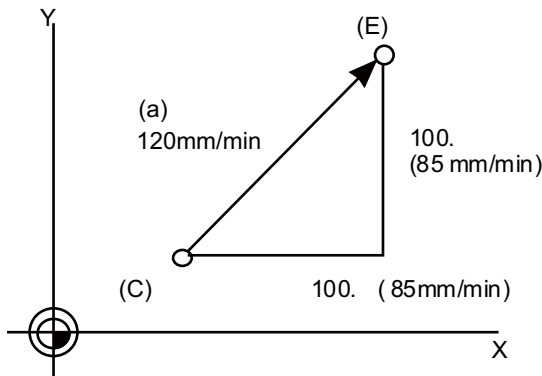
Linear interpolation is executed by the above command at the f1 feed rate. The tool path takes the shortest distance to the end point in the form of a straight line.

For details on the f1 command values for NC, refer to the section entitled "Cutting Feed Rate".

Since the actual cutting feed rate depends on the machine, refer to the specifications of the machine concerned.

(Example)

G01 G91 X100. Y100. F120 ;



The cutting feed rate command moves the tool in the vector direction. The component speeds of each axis are determined by the proportion of respective command values.

- (a) Cutting feed rate
- (C) Current position
- (E) End point

- (1) The number of axes which can be commanded simultaneously depends on the specifications (number of simultaneously controlled axes). The axes can be used in any combination within this range.
- (2) The feed rate is controlled so that it does not exceed the cutting feed rate clamp of each axis.
- (3) 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)

M system	○
L system	○

(1) 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 coordinate
Ii1,Jj1 : Arc center
Ff1 : Feed rate

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 specified by G02 or G03.

G02: Clockwise (CW)

G03: Counterclockwise (CCW)

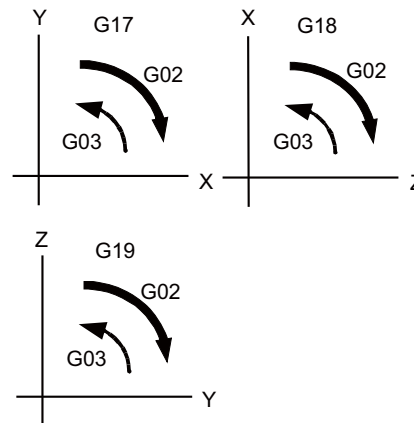
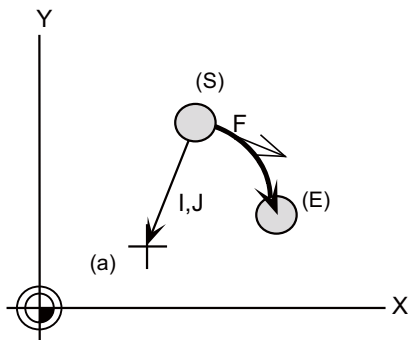
The plane is selected by G17, G18 or G19.

G17: XY plane

G18: ZX plane

G19: YZ plane

(Example) See below for examples of circular commands.



(a) Center
(E) End point
(S) Start point

- (a) The axes that can be commanded simultaneously are the two axes for the selected plane.
- (b) The feed rate is controlled so that the tool always moves at a speed along the circumference of the circle.
- (c) Circular interpolation can be commanded within a range extending from 0° to 360° .
- (d) The max. value of the radius can be set up to six digits above the decimal point.

(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.

(Note 2) The axes configuring a plane can be designated by parameters. Refer to the section entitled "Plane Selection".

(2) 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 coordinate
Rr1          : Arc radius
Ff1          : Feed rate
```

G02 or G03 is used to designate the direction of the arc rotation.

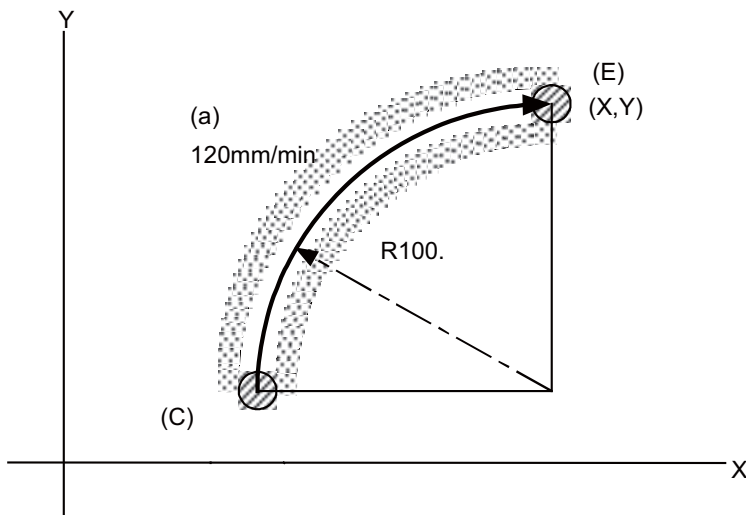
The arc plane is designated by G17, G18 or G19.

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°.

(Example)

```
G02 G91 X100. Y100. R100. F120 ;
```



- (a) Feed rate
- (C) Current position (arc start point)
- (E) Arc end point

- (a) The axes that can be commanded simultaneously are the two axes for the selected plane.
- (b) The feed rate is controlled so that the tool always moves at a speed along the circumference of the circle.

(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

M system	○
L system	○

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 be exercised to machine large-diameter screws or 3-dimensional cams.

G17 G02(G03) Xx1 Yy1 Zz1 Ii1 Jj1 Pp1 Ff1 ; (Specify arc center)
G17 G02(G03) Xx1 Yy1 Zz1 Rr1 Ff1 ; (Specify arc radius "R")

G17 : Arc plane
 G02,G03 : Arc rotation direction
 Xx1,Yy1 : End point coordinate values for arc
 Zz1 : End point coordinate value of linear axis
 Ii1,Jj1 : Arc center coordinate values
 Pp1 : Number of pitches
 Ff1 : Feed rate
 Rr1 : Arc radius

- (1) The arc plane is designated by G17, G18 or G19.
- (2) G02 or G03 is used to designate the direction of the arc rotation.
- (3) Absolute or incremental values can be assigned for the arc end point coordinates and the end point coordinates of the linear axis, but incremental values must be assigned for the arc center coordinates.
- (4) The linear interpolation axis is the other axis which is not included in the plane selection.
- (5) Command the speed in the component direction that represents all the axes combined for the feed rate. Pitch I1 is obtained by the formula below.

$$I1 = z1 / ((2\pi * p1 + \theta) / 2\pi)$$

$$\theta = \theta e - \theta s = \arctan(ye/xs) - \arctan(ys/xs)$$

Where xs, ys are the start point coordinates ($0 \leq \theta < 2\pi$)

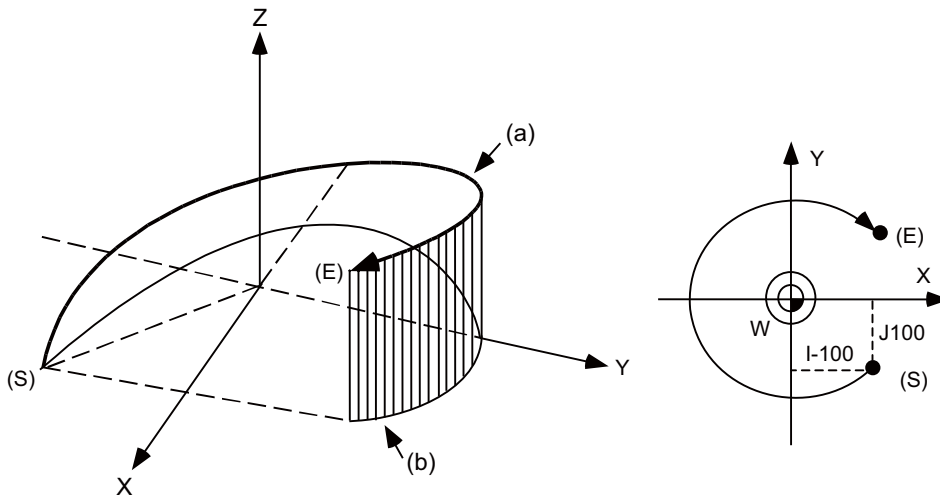
xe, ye are the end point coordinates

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.

(Example)

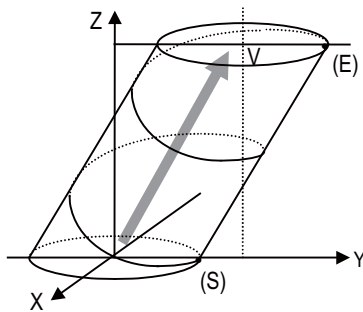
```
G91 G17 G02 X0. Y200. Z100. I-100. J100. F120 ;
```



- (a) Command program path
- (b) XY plane projection path in command program
- (S) Start point
- (E) End point

(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 shown in the figure on the right. In other words, linear interpolation of the Z and V axes is carried out in synchronization with the circular interpolation on the XY plane.



- (E) Start point
- (S) End point



Feed

4.1 Feed Rate

4.1.1 Rapid Traverse Rate (m/min)

M system	1000
L system	1000

[M system]

The rapid traverse rate can be set independently for each axis by the parameter.

The rapid traverse rate is effective for G00, G27, G28, G29, G30 and G60 commands.

Override can be applied to the rapid traverse rate using the external signal supplied.

Rapid traverse rate setting range

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

Least command increment B : 0.001 mm (0.0001inch)

Least command increment C : 0.0001mm (0.00001inch)

[L system]

The rapid traverse rate can be set independently for each axis by the parameter.

The rapid traverse rate is effective for G00, G27, G28, G29, G30 and G53 commands.

Override can be applied to the rapid traverse rate using the external signal supplied.

Rapid traverse rate setting range

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

Least command increment B : 0.001mm (0.0001inch)

Least command increment C : 0.0001mm (0.00001inch)

4.1.2 Cutting Feed Rate (m/min)

M system	1000
L system	1000

[M system]

This function specifies the feed rate 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 feed rate modal value is cleared to zero only when the power is turned ON.

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

Cutting feed rate setting range

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

Least command increment B : 0.001mm (0.0001inch)

Least command increment C : 0.0001mm (0.00001inch)

Least command increment D : 0.00001mm (0.000001inch)

Least command increment E : 0.000001mm (0.0000001inch)

- G code command for which the cutting feed rate 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 feed rate 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 feed rate modal is cleared to zero only when the power is turned ON.

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

Cutting feed rate setting range

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

Least command increment B : 0.001mm (0.0001inch)

Least command increment C : 0.0001mm (0.00001inch)

Least command increment D : 0.00001mm (0.000001inch)

Least command increment E : 0.000001mm (0.0000001inch)

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

4.1.3 Manual Feed Rate (m/min)

M system	1000
L system	1000

The manual feed rates are designated as the feed rate in the jog mode or incremental feed mode for manual operation and the feed rate during dry run ON for automatic operation. The manual feed rates are set with external signals. The manual feed rate 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

M system	<input type="radio"/>
L system	<input type="radio"/>

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

Automatic operation	Cutting feed rate	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 Feed Rate Input Methods

4.2.1 Feed per Minute (Asynchronous Feed)

M system	<input type="radio"/>
L system	<input type="radio"/>

[M system]

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

Metric input (mm)

Input Setting unit	B(0.001mm)	C(0.0001mm)
Command Address	F(mm/min)	F(mm/min)
Minimum command unit	1(=1.00) (1.=1.00)	1(=1.000) (1.=1.000)
Command range	0.01 - 1000000.00	0.001 - 1000000.000

Inch input (inch)

Input Setting unit	B(0.0001inch)	C(0.00001inch)
Command Address	F(inch/min)	F(inch/min)
Minimum command unit	1(=1.000) (1.=1.000)	1(=1.0000) (1.=1.0000)
Command range	0.001 - 100000.000	0.0001 - 100000.0000

[L system]

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

Metric input (mm)

Input Setting unit	B(0.001mm)	C(0.0001mm)
Command Address	F(mm/min)	F(mm/min)
Minimum command unit	1(=1.000) (1.=1.000)	1(=1.0000) (1.=1.0000)
Command range	0.001 - 1000000.000	0.0001 - 1000000.0000

Inch input (inch)

Input Setting unit	B(0.0001inch)	C(0.00001inch)
Command Address	F(inch/min)	F(inch/min)
Minimum command unit	1(=0.0100) (1.=1.0000)	1(=0.01000) (1.=1.00000)
Command range	0.0001 - 100000.0000	0.00001 - 100000.00000

4.2.2 Feed per Revolution (Synchronous Feed)

M system	<input type="radio"/>
L system	<input type="radio"/>

By issuing the G95 command, the commands from that block are issued directly by the numerical value following F as the feed rate per spindle revolution (mm/revolution or inch/revolution).

The least command increment and command range of the feed rate designation F are as follows.

[M system]

Metric input (mm)

Input Setting unit	B(0.001mm)	C(0.0001mm)
Command Address	F(mm/rev)	F(mm/rev)
Minimum command unit	1(=0.001) (1.=1.0)	1(=0.0001) (1.=1.0)
Command range	0.001 - 999.999	0.0001 - 999.9999

Inch input (inch)

Input Setting unit	B(0.0001inch)	C(0.00001inch)
Command Address	F(inch/rev)	F(inch/rev)
Minimum command unit	1(=0.00001) (1.=1.00)	1(=0.000001) (1.=1.00)
Command range	0.00001 - 99.99999	0.000001 - 99.999999

[L system]

Metric input (mm)

Input Setting unit	B(0.001mm)	C(0.0001mm)
Command Address	F(mm/rev)	F(mm/rev)
Minimum command unit	1(=0.0001) (1.=1.00)	1(=0.00001) (1.=1.00)
Command range	0.0001 - 999.9999	0.00001 - 999.99999

Inch input (inch)

Input Setting unit	B(0.0001inch)	C(0.00001inch)
Command Address	F(inch/rev)	F(inch/rev)
Minimum command unit	1(=0.000001) (1.=1.000)	1(=0.0000001) (1.=1.000)
Command range	0.000001 - 99.999999	0.0000001 - 99.9999999

4.2.4 F 1-digit Feed

M system	<input type="radio"/>
L system	<input type="radio"/>

The feed rate 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 feed rate has been issued as an F 1-digit command, the feed rate can be increased or reduced by turning the manual handle.

The feed rate cannot be changed by the 2nd and 3rd handles.

(1) 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 pulses (\pm)

FM : F1 to F5 upper limit (parameter setting)

K : Speed variation constant (parameter setting)

(Example) When the feed rate is to be increased or reduced by 10 mm/min per manual handle scale increment if FM is 3600 mm/min, then:

$$\Delta F = 10 = 1 \times \frac{3600}{K} \quad \text{Therefore, } K = 360.$$

(2) Conditions under which F1-digit feed is valid

- (a) The automatic operation must be selected.
- (b) Automatic start must be underway.
- (c) Cutting feed must be underway, and the F 1-digit feed rate must be designated.
- (d) The F 1-digit valid parameter must be ON.
- (e) The F 1-digit feed rate change valid signal must be ON.
- (f) A dry run must not be in progress.
- (g) Machine lock must not be activated.

4.2.5 Manual Speed Command

M system	○
L system	○

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.

4.3 Override

4.3.1 Rapid Traverse Override

M system	○
L system	○

(1) 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.

(2) 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.

(Note 2) A PLC must be built into the unit for type 2.

4.3.2 Cutting Feed Override

M system	○
L system	○

(1) 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.

(2) 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) A PLC must be built into the unit for type 2.

4.3.3 2nd Cutting Feed Override

M system	○
L system	○

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) A PLC must be built into the unit for this function.

4.3.4 Override Cancel

M system	○
L system	○

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.

(Note 2) When the cutting feed override or second cutting feed override is 0%, the 0% override takes precedence and the override is not canceled.

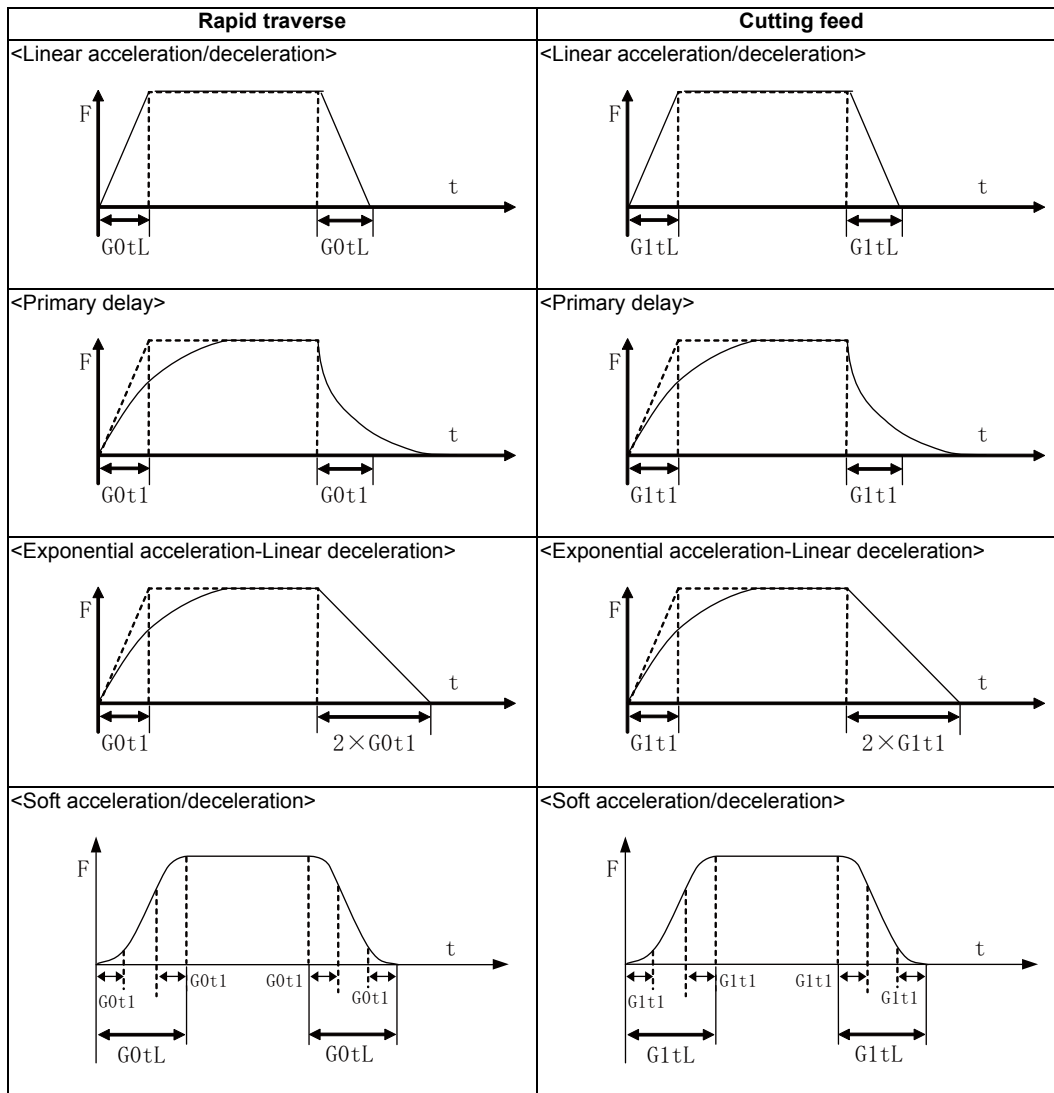
(Note 3) The override cancel signal is not valid for rapid traverse.

4.4 Acceleration/Deceleration

4.4.1 Automatic Acceleration/Deceleration after Interpolation

M system	○
L system	○

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.

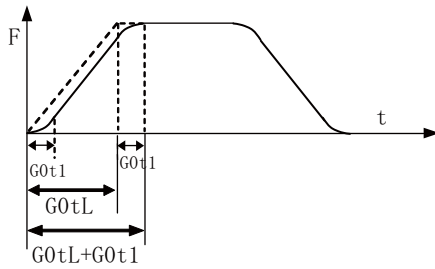


- G0tL:G0 time constant(linear)
- G0t1:G0 time constant(primary delay)/The 2nd step time constant of the soft acceleration/deceleration
- G1tL:G1 time constant(linear)
- G1t1:G1 time constant(primary delay)/The 2nd step time constant of the soft acceleration/deceleration

(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.
 It is invalid to G31 and handle feed.

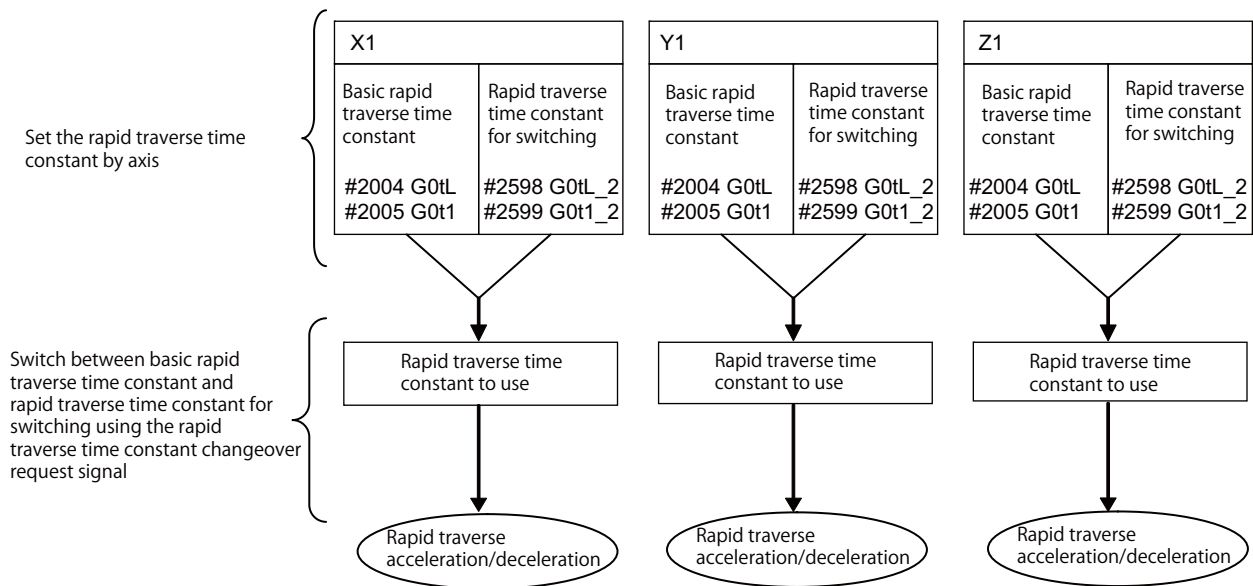
(Note 2) Acceleration/deceleration in handle feed mode is usually performed according to the acceleration/deceleration pattern for cutting feed. However, a parameter can be specified to select a pattern with no acceleration/deceleration (step).

(Note 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 changeover request signal.



4.4.2 Rapid Traverse Constant Inclination Acceleration/Deceleration

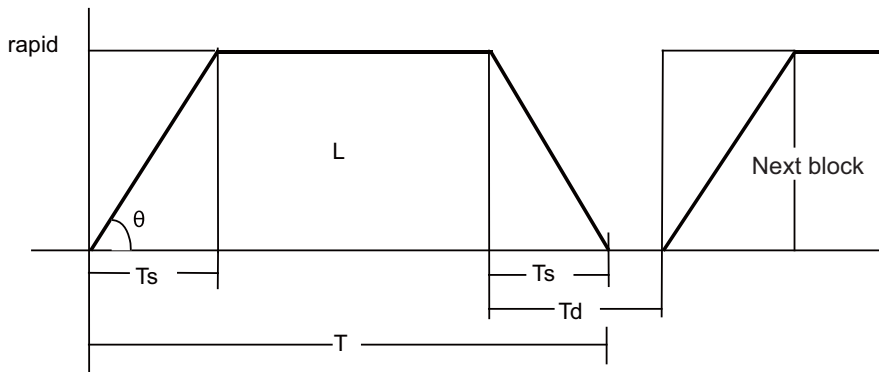
M system	○
L system	○

This function performs acceleration and deceleration at a constant inclination during linear acceleration/ deceleration in the rapid traverse mode. Compared to the method of acceleration/deceleration after interpolation, the constant inclination acceleration/deceleration method makes for improved cycle time.

Rapid traverse constant inclination acceleration/deceleration are valid only for a rapid traverse command. Also, this function is effective only when the rapid traverse command acceleration/deceleration mode is linear acceleration and linear deceleration.

The acceleration/deceleration patterns in the case where rapid traverse constant inclination acceleration/ deceleration are performed are as follows.

(1) When the interpolation distance is long enough for the rapid traverse rate to be achieved



$$T = \frac{L}{\text{rapid}} + T_s$$

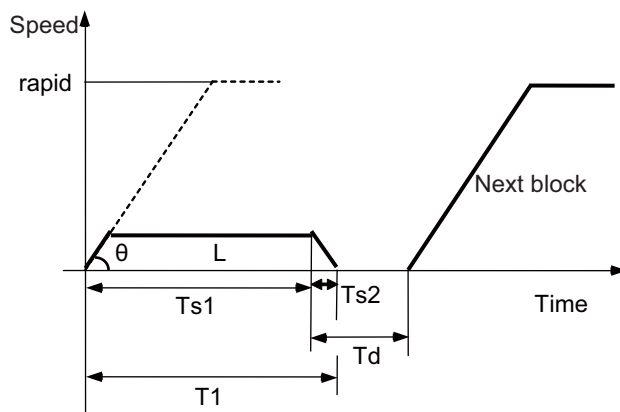
$$T_d = T_s + (0 \text{ to } 14 \text{ ms})$$

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

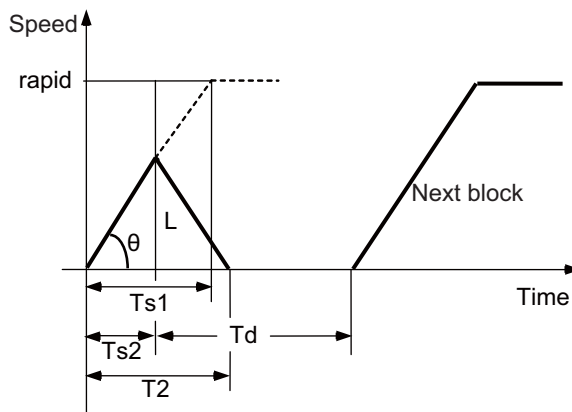
- rapid : Rapid traverse rate
- Ts : Acceleration/deceleration time constant
- Td : Command deceleration check time
- θ : Acceleration/deceleration inclination
- T : Interpolation time
- L : Interpolation distance

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

In case of time-constant acceleration/deceleration:



In case of inclination-constant acceleration/deceleration:



$$T1 = Ts1 + Ts2$$

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

$$Td = T2 / 2 + (0 \text{ to } 14 \text{ ms})$$

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

rapid: Rapid traverse rate (Axis specification parameter #2001 rapid)

Ts1: Acceleration/deceleration time (Axis specification parameter #2004 G0tL)

Ts2: Acceleration/deceleration time to reach the maximum speed

Td: Command deceleration check time

θ : Acceleration/deceleration inclination

T1: Interpolation time (time-constant acceleration/deceleration)

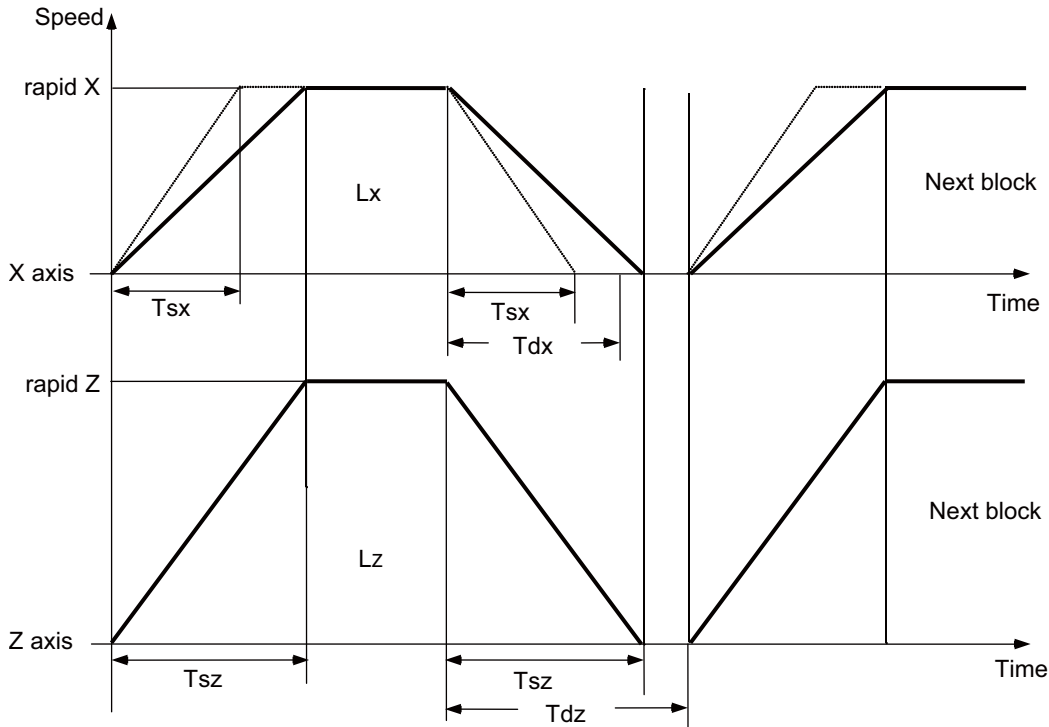
T2: Interpolation time (inclination-constant acceleration/deceleration)

L: Interpolation distance

The time required to perform a command deceleration check during rapid traverse constant inclination acceleration/deceleration is the longest value among the rapid traverse deceleration check times determined for each axis by the rapid traverse rate of commands executed simultaneously, the rapid traverse acceleration/deceleration time constant, and the interpolation distance, respectively.

(3) 2-axis simultaneous interpolation (When linear interpolation is used, $T_{sx} < T_{sz}$, and $L_x \neq L_z$)

When 2-axis simultaneous interpolation (linear interpolations) is performed during rapid traverse constant inclination acceleration and deceleration, the acceleration (deceleration) time is the longest value of the acceleration (deceleration) times determined for each axis by the rapid traverse rate of commands executed simultaneously, the rapid traverse acceleration and deceleration time constant, and the interpolation distance, respectively. Consequently, linear interpolation is performed even when the axes have different acceleration and deceleration time constants.



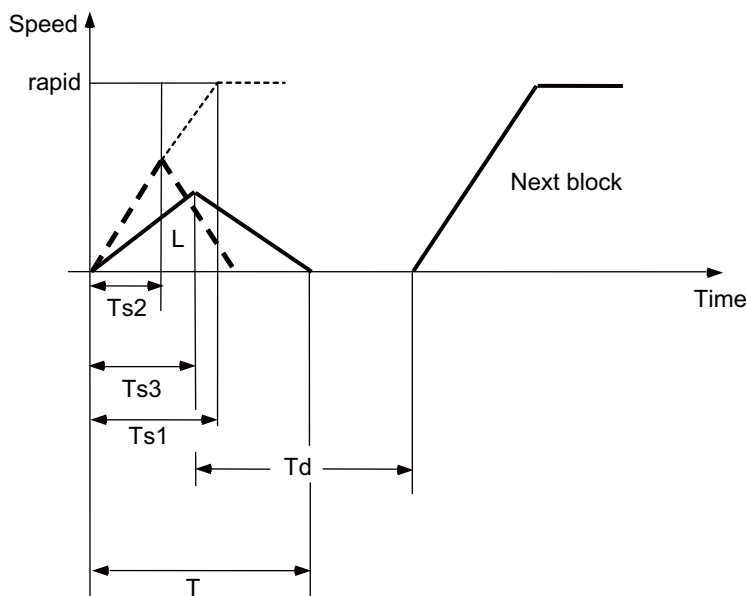
When T_{sz} is greater than T_{sx} ,
 T_{dz} is also greater than T_{dx} , and
 $T_d = T_{dz}$ in this block.

- Tsx: X axis acceleration/deceleration time
- Tsz: Z axis acceleration/deceleration time
- Tdx: X axis commanded deceleration check time
- Tdz: Z axis commanded deceleration check time
- Lx : X axis interpolation distance
- Lz : Z axis interpolation distance

The program format of G0 (rapid traverse command) when rapid traverse constant inclination acceleration/ deceleration are executed is the same as when this function is invalid (time constant acceleration/deceleration). This function is valid only for G0 (rapid traverse).

(4) **When the interpolation distance is so short that the acceleration/deceleration time is shorter than the minimum time constant for constant inclination acceleration/deceleration**

If a minimum time constant for inclination-constant acceleration/deceleration by the parameter, acceleration/deceleration speed is adjusted to prevent the acceleration/deceleration time calculated by interpolation distance from going below the minimum time constant.



$$T = 2 \times Ts2$$

$$Td = \frac{T}{2} + (0 \text{ to } 14 \text{ ms})$$

rapid: Rapid traverse speed (Axis specification parameter #2001 rapid)

Ts1: Acceleration/deceleration time (Axis specification parameter #2004 G0tL)

Ts2: Acceleration/deceleration time to reach the maximum speed

Ts3: Minimum time for inclination-constant acceleration/deceleration (Axis specification parameter #2198 G0tMin)

Td: Command deceleration check time

T: Interpolation time

L: Interpolation distance

4.5 Thread Cutting

4.5.1 Thread Cutting (Lead/Thread Number Designation)

M system	○
L system	○

The thread cutting with designated lead can be performed. Inch threads are cut by designating the number of threads per inch with the E address.

(1) Lead designation

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

G33 Zz1/Ww1 Xx1/Uu1 Qq1 Ff1/Ee1 ;	
G33	: Thread cutting command
Zz1/Ww1,Xx1/Uu1	: Thread end point coordinates
Qq1	: Shift angle at start of thread cutting (0.000 to 360.000°)
Ff1	: Thread lead (normal lead threads)
Ee1	: Thread lead (precise lead threads)

(2) Thread number designation

Inch threads are cut by designating the number of threads per inch with the E address.

Whether the E command is a thread number designation or lead designation is selected with the parameters.

G33 Zz1/Ww1 Xx1/Uu1 Qq1 Ee1 ;	
G33	: Thread cutting command
Zz1/Ww1,Xx1/Uu1	: Thread end point coordinates
Qq1	: Shift angle at start of thread cutting (0.000 to 360.000°)
Ee1	: Thread number per inch

[M system]

Thread cutting metric input

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

Thread cutting inch input

Input setting unit	B(0.0001inch)			C(0.00001inch)		
	F(inch/rev)	E(inch/rev)	E (ridges/inch)	F(inch/rev)	E(inch/rev)	E (ridges/inch)
Least Command Increments	1(=1.0000) (1.=1.0000)	1(=1.00000) (1.=1.00000)	1(=1.000) (1.=1.000)	1(=1.00000) (1.=1.00000)	1(=1.000000) (1.=1.000000)	1(=1.0000) (1.=1.0000)
Command range	0.0001 - 39.3700	0.00001 - 39.37007	0.025 - 9999.999	0.00001 - 39.37007	0.000001 - 39.370078	0.0254 - 9999.9999

[L system]

Thread cutting metric input

Input setting unit	B (0.000mm)			C (0.0001mm)		
Command address	F (mm/rev)	E (mm/rev)	E (ridges/inch)	F (mm/rev)	E (mm/rev)	E (ridges/inch)
Least Command Increments	1(=1.0000) (1.=1.0000)	1(=1.00000) (1.=1.00000)	1(=1.00) (1.=1.00)	1(=1.00000) (1.=1.00000)	1(=1.000000) (1.=1.000000)	1(=1.000) (1.=1.000)
Command range	0.0001 - 999.9999	0.00001 - 999.99999	0.03 - 999.99	0.00001 - 999.99999	0.000001 - 999.999999	0.026 - 222807.017

Thread cutting inch input

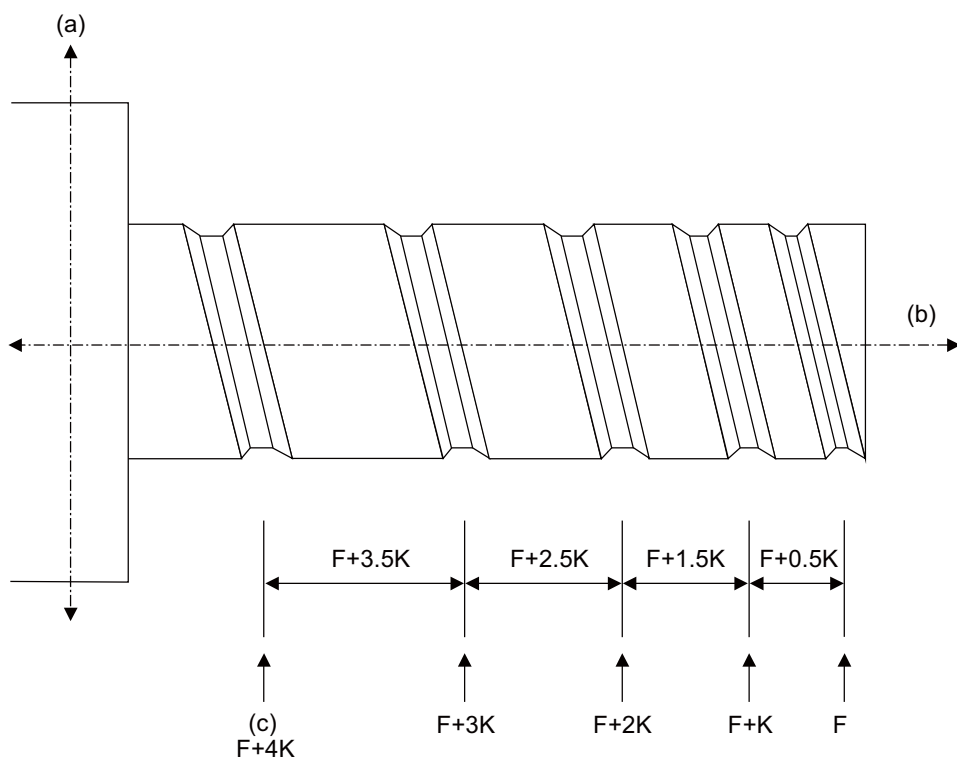
Input setting unit	B (0.0001inch)			C (0.00001inch)		
Command address	F (inch/rev)	E (inch/rev)	E (ridges/inch)	F (inch/rev)	E (inch/rev)	E (ridges/inch)
Least Command Increments	1(=1.00000) (1.=1.00000)	1(=1.000000) (1.=1.000000)	1(=1.000) (1.=1.000)	1(=1.000000) (1.=1.000000)	1(=1.0000000) (1.=1.0000000)	1(=1.0000) (1.=1.0000)
Command range	0.00001 - 39.37007	0.000001 - 39.370078	0.025 - 9999.999	0.000001 - 39.370078	0.0000001 - 39.3700787	0.0254 - 9999.9999

4.5.2 Variable Lead Thread Cutting

M system	—
L system	○

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

```
G34 Xx1/Uu1 Zz1/Ww1 Ff1/Ee1 Kk1 ;
G34          : Variable lead thread cutting command
Xx1/Uu1      : Thread end point X coordinate
Zz1/Ww1      : Thread end point Z coordinate
Ff1/Ee1      : Thread's basic lead
Kk1          : Lead increment/decrement amount per thread rotation
```



- (a) Non-lead axis
- (b) Lead axis
- (c) Lead speed

4.5.3 Synchronous Tapping

4.5.3.1 Synchronous Tapping Cycle

M system	O
L system	O

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.

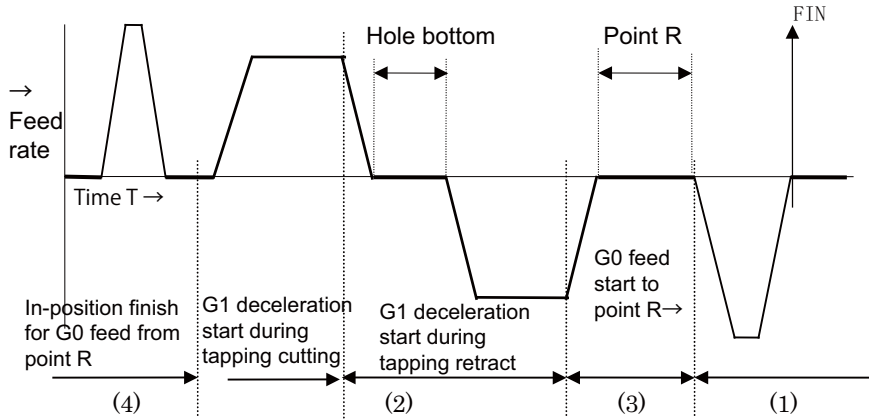
(1) Tapping pitch assignment

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 speed
,Ss2	: Rotation speed of spindle 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

(2) 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 speed
,Ss2	: Rotation speed of spindle 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.

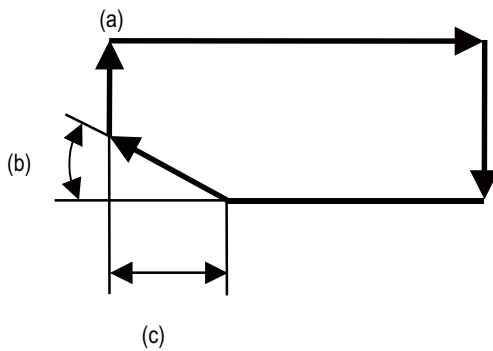


- (1) Section where in-position check is performed using servo in-position width
- (2) Section where in-position check is performed using in-position width for tapping
- (3) Section where in-position check is performed using in-position width for cutting feed (G1, G2, G3)
- (4) Section where in-position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

4.5.4 Chamfering

M system	—
L system	○

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



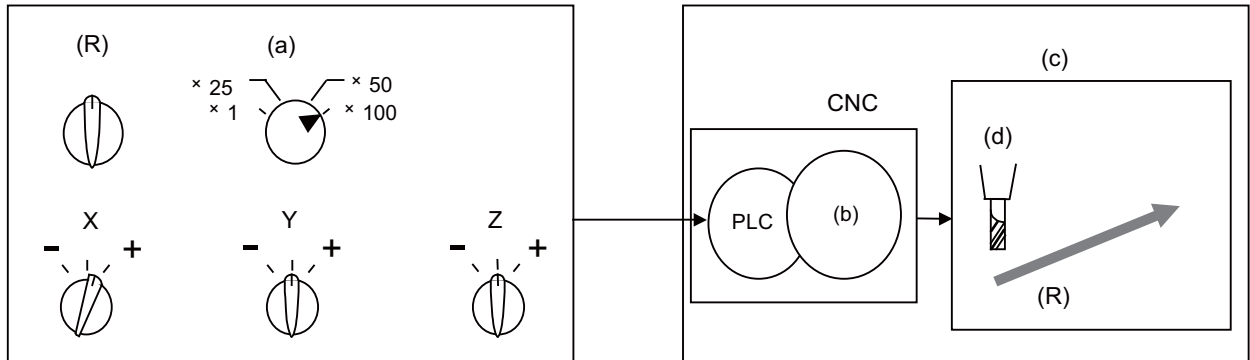
- (a) Thread cutting cycle
- (b) Chamfer angle
- (c) Chamfer amount

4.6 Manual Feed

4.6.1 Manual Rapid Traverse

M system	<input type="radio"/>
L system	<input type="radio"/>

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.

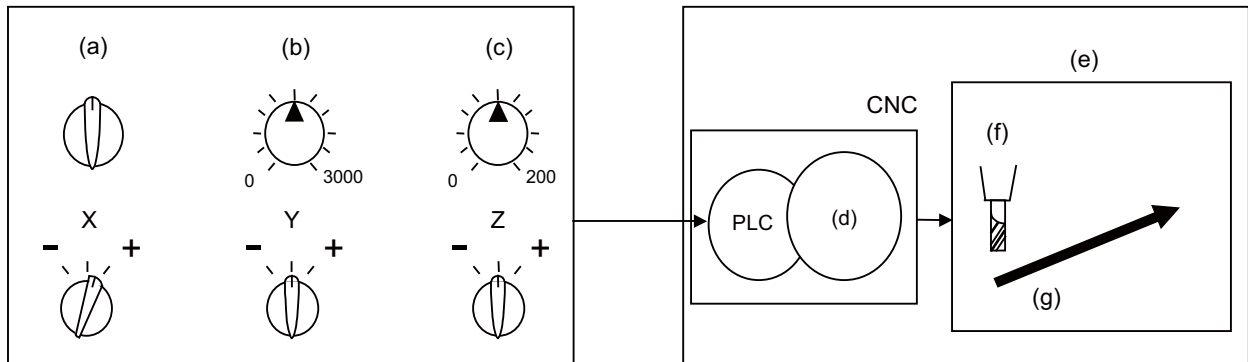


- (a) Rapid traverse override
- (b) Axis movement control
- (c) Machine tool
- (d) Tool
- (R) Rapid traverse

4.6.2 Jog Feed

M system	<input type="radio"/>
L system	<input type="radio"/>

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.

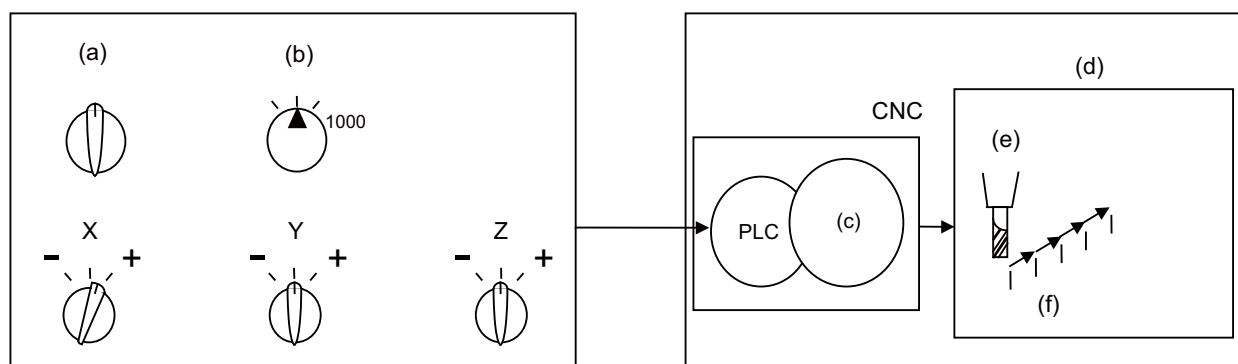


- (a) Jog
- (b) Feed rate
- (c) Override
- (d) Axis movement control
- (e) Machine tool
- (f) Tool
- (g) Manual cutting feed

4.6.3 Incremental Feed

M system	<input type="radio"/>
L system	<input type="radio"/>

When the incremental feed mode is selected, the tool can be operated by an amount equivalent to the designated amount (incremental value) 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.



- (a) Incremental
- (b) Scale factor
- (c) Axis movement control
- (d) Machine tool
- (e) Tool
- (f) Step feed

4.6.4 Handle Feed

M system	<input type="radio"/>
L system	<input type="radio"/>

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 X1, X10, X100, X1000 or arbitrary value.

If the least command increment is 10nm or 1nm, the scale can be selected from X5000, X10000 or X100000, as well. Individual axes can be moved in very small amounts either separately or simultaneously 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 can be used with the Mitsubishi CNC.

4.6.5 Manual Feed Rate B

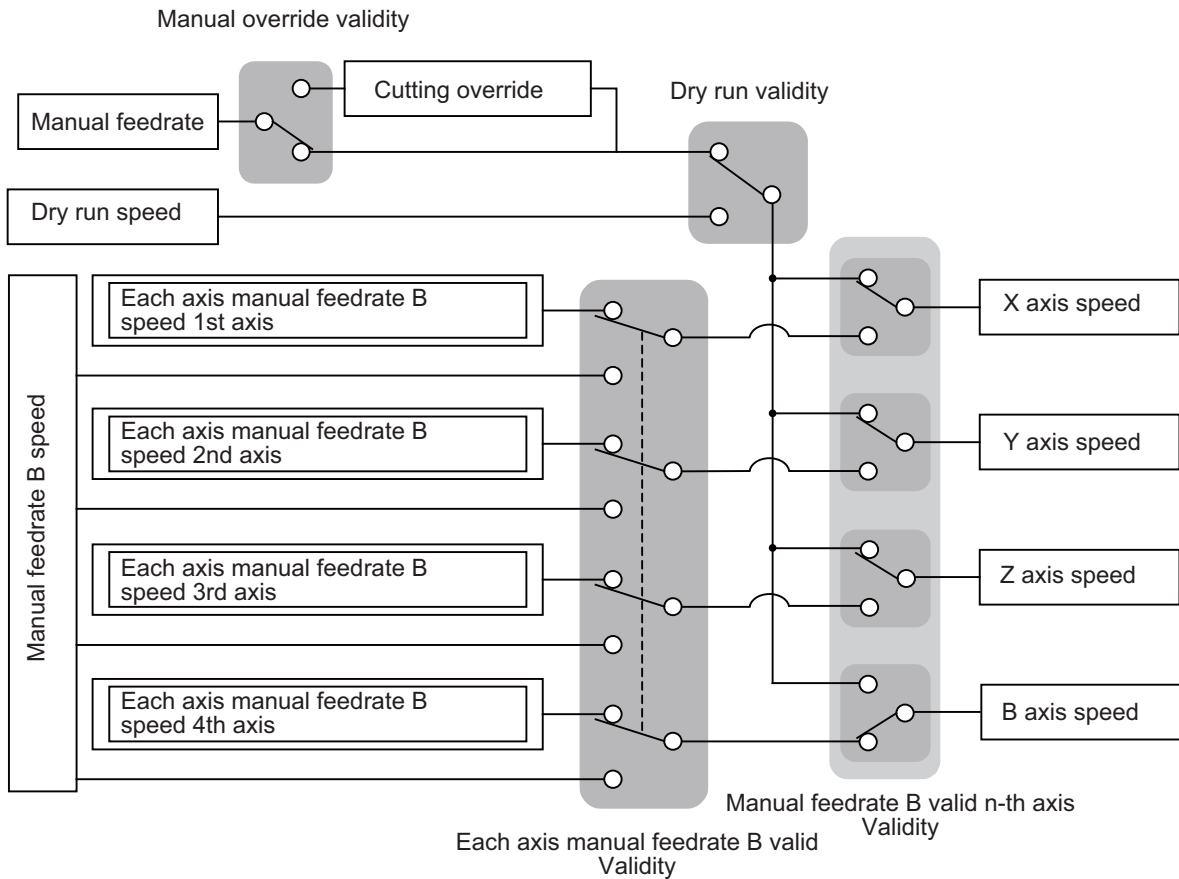
M system	○
L system	○

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

4.7 Dwell

4.7.1 Dwell (Time-based Designation)

M system	○
L system	○

The G04 command temporarily stops the machine movement and sets the machine standby status for the time designated in the program.

(1) M system

G04 Xx1 ; or G04 Pp1 ;	
G04	: Dwell
Xx1,Pp1	: Dwell time

The time-based dwell can be designated in the range from 0.001 to 99999.999 seconds.
(The input command increment for the dwell time depends on the parameter.)

(2) L system

(G94) G04 Xx1/Uu1 ; or (G94) G04 Pp1 ;	
G94	: Asynchronous
G04	: Dwell command
Xx1,Uu1,Pp1	: Dwell time

The time-based dwell can be designated in the range from 0.001 to 99999.999 seconds. (The input command increment for the dwell time depends on the parameter.)

U address can be used in the L system only.

5

Program Memory/Editing

5.1 Memory Capacity

Machining programs are stored in the NC memory and a memory card.

When using a memory card, mass-editing 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)

5.1.1.5 230kB [600 m] (400 Programs)

M system	<input type="radio"/>
L system	<input type="radio"/>

5.2 Editing

5.2.1 Program Editing

M system	<input type="radio"/>
L system	<input type="radio"/>

The following editing functions are possible.

(1) Program erasing

- (a) Machining programs can be erased individually or totally.

(2) Program filing

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

(3) Program copying

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

(4) Program editing

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

(5) Mass editing

With mass-editing, up to 10 MB can be edited.

The specification and restrictions are different from those with the regular editing.

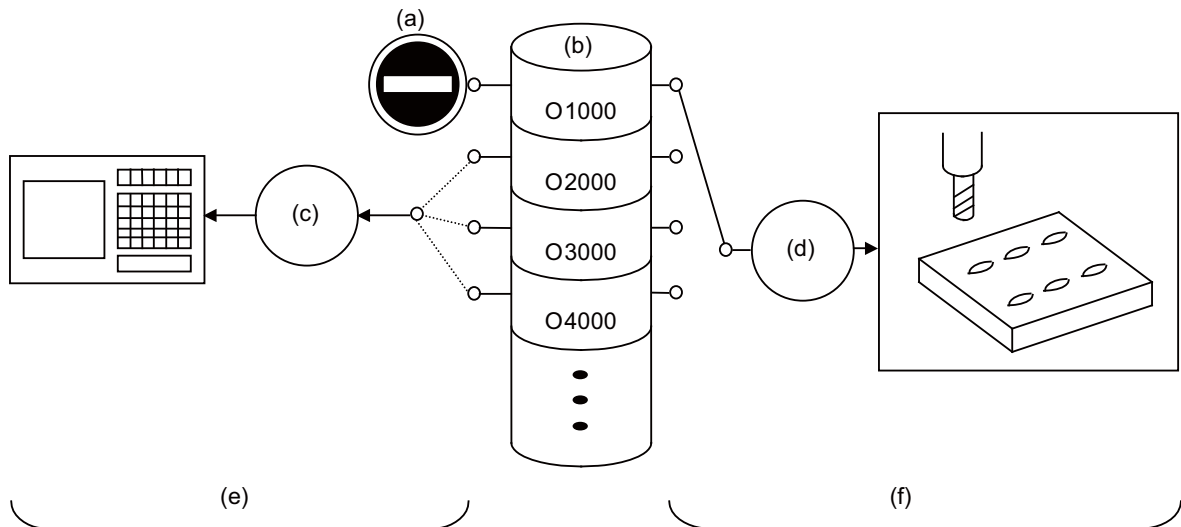
Mass-editing is applied when all of the following conditions are satisfied.

- When the storage destination for the program to be opened is a memory card.
- When a file size is 0.5 MB or larger.

5.2.2 Background Editing

M system	<input type="radio"/>
L system	<input type="radio"/>

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



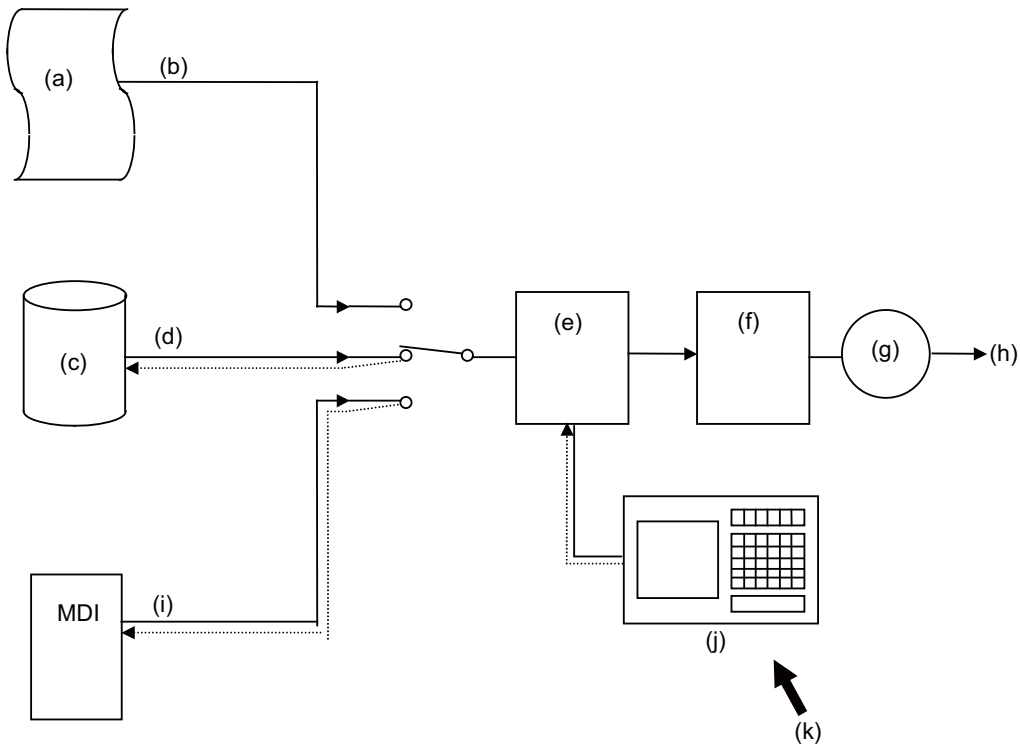
- (a) Prohibited
- (b) Program registered in memory
- (c) Editing
- (d) Memory operation
- (e) Program editing
- (f) Machining with memory operation

- (1) 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.
- (2) 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.
- (3) The machining program will not be searched as the operation target even when searched in the edit screen.

5.2.3 Buffer Correction

M system	○
L system	○

During automatic operation (including memory, tape, memory card) or MDI operation, this function initiates single block stop and enables the next command to be corrected or changed.
 Only memory 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.



- (a) Tape
- (b) Tape mode
- (c) Memory, memory card
- (d) Memory mode
- (e) Pre-read block
- (f) Execution block
- (g) NC operation
- (h) Machine control
- (i) MDI mode
- (j) Setting and display unit
- (k) Buffer correction

6

Operation and Display

6.1 Structure of Operation/Display Panel

The setting and display unit is configured of the setting section and keyboard section.
Refer to "General Specifications" for details.

6.1.1 Color Display (8.4-type LCD TFT)

M system	<input type="checkbox"/>
L system	<input type="checkbox"/>

6.2 Operation Methods and Functions

6.2.1 Operation Input

M system	<input type="radio"/>
L system	<input type="radio"/>

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. The operation results appear when the INPUT key is pressed. If the INPUT key is pressed again, the data is processed and displayed 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 Value Setting

M system	<input type="radio"/>
L system	<input type="radio"/>

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

6.2.6 Menu List

M system	<input type="radio"/>
L system	<input type="radio"/>

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

M system	<input type="radio"/>
L system	<input type="radio"/>

The screen display changes when the screen mode selection switch is changed.

The details corresponding to the operation mode are displayed.

6.2.10 Screen Saver

M system	<input type="radio"/>
L system	<input type="radio"/>

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.

6.2.11 Parameter/Operation Guidance

M system	<input type="radio"/> (*2)
L system	<input type="radio"/> (*2)

*2: Guidance data add-on is needed in case of FCU7-MU558-001.

The parameter/operation guidance function displays the details of the parameters or the operation methods according to the state of the screen currently being displayed. The operation guidance can also be selected from the Contents and displayed.

If the ? key is pressed on any screen, the parameter/operation guidance window will open. If a pop-up window other than the parameter/operation guidance window is opened, the parameter/operation guidance window will open over the currently opened pop-up window.

6.2.12 Alarm Guidance

M system	<input type="radio"/> (*2)
L system	<input type="radio"/> (*2)

*2: Guidance data add-on is needed in case of FCU7-MU558-001.

Guidance is displayed 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.

6.2.15 Screen Capture

M system	<input type="radio"/>
L system	<input type="radio"/>

This function allows to output a bitmap file of a screen displayed on the setting and display unit.

6.2.16 User Selectable Menu Configuration

M system	<input type="radio"/>
L system	<input type="radio"/>

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

6.2.20 MTB Selectable Menu Configuration

M system	<input type="radio"/>
L system	<input type="radio"/>

Menu items on the "Monitor", "Setup", and "Edit" screens (of MITSUBISHI 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.

6.3 Display Methods and Contents

6.3.1 Status Display

M system	<input type="radio"/>
L system	<input type="radio"/>

The status of the program now being executed is indicated.

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

(*) 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 servomotor.

6.3.2 Clock Display

M system	<input type="radio"/>
L system	<input type="radio"/>

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

M system	<input type="radio"/>
L system	<input type="radio"/>

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

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

6.3.4 Setup Screen Display

M system	<input type="radio"/>
L system	<input type="radio"/>

Tool/workpiece related settings, user parameter settings, MDI editing, counter setting and manual numeric command issuing can be carried out on the Preparation screen.

6.3.5 Edit Screen Display

M system	<input type="radio"/>
L system	<input type="radio"/>

Machining program editing (addition, deletion, change) and checking, simple program creation, and machining program input/output can be carried out on the Edit screen.

6.3.6 Diagnosis Screen Display

M system	<input type="radio"/>
L system	<input type="radio"/>

The following operations related to the CNC diagnosis can be carried out on the Diagnosis screen.

- (1) Display of hardware and software configuration
- (2) Display of CNC options
- (3) Diagnosis of PLC interface
- (4) Display of drive unit information
- (5) Display of alarm message / alarm history list etc.

6.3.7 Maintenance Screen Display

M system	<input type="radio"/>
L system	<input type="radio"/>

Parameter setting and display, and NC data input/output, etc., can be carried out on the Maintenance screen.

6.3.8 Additional Language

A language of choice can be used for display by parameter setting.

(Note) Guidance language data is separately provided. Up to two languages besides English can be additionally selected.

6.3.8.1 Japanese

M system	<input type="checkbox"/>
L system	<input type="checkbox"/>

6.3.8.2 English

M system	<input type="radio"/>
L system	<input type="radio"/>

6.3.8.3 German

M system	<input type="checkbox"/>
L system	<input type="checkbox"/>

6.3.8.4 Italian

M system	<input type="checkbox"/>
L system	<input type="checkbox"/>

6.3.8.5 French

M system	<input type="checkbox"/>
L system	<input type="checkbox"/>

6.3.8.6 Spanish

M system	<input type="checkbox"/>
L system	<input type="checkbox"/>

6.3.8.7 Chinese

6.3.8.7.1 Chinese (Traditional Chinese Characters)

M system	<input type="checkbox"/>
L system	<input type="checkbox"/>

6.3.8.7.2 Chinese (Simplified Chinese Characters)

M system	<input type="checkbox"/>
L system	<input type="checkbox"/>

6.3.8.8 Korean

M system	<input type="checkbox"/>
L system	<input type="checkbox"/>

6.3.8.9 Portuguese

M system	<input type="checkbox"/>
L system	<input type="checkbox"/>

6.3.8.10 Hungarian

M system	<input type="checkbox"/>
L system	<input type="checkbox"/>

6.3.8.11 Dutch

M system	<input type="checkbox"/>
L system	<input type="checkbox"/>

6.3.8.12 Swedish

M system	<input type="checkbox"/>
L system	<input type="checkbox"/>

6.3.8.13 Turkish

M system	<input type="checkbox"/>
L system	<input type="checkbox"/>

6.3.8.14 Polish

M system	<input type="checkbox"/>
L system	<input type="checkbox"/>

6.3.8.15 Russian

M system	<input type="checkbox"/>
L system	<input type="checkbox"/>

6.3.8.16 Czech

M system	<input type="checkbox"/>
L system	<input type="checkbox"/>

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 (Including User Macros and Fixed Cycle Macros)

M system	<input type="radio"/>
L system	<input type="radio"/>

7.1.2 Tool Offset Data Input / Output

M system	<input type="radio"/>
L system	<input type="radio"/>

7.1.3 Common Variable Input / Output

M system	<input type="radio"/>
L system	<input type="radio"/>

7.1.4 Parameter Input / Output

M system	<input type="radio"/>
L system	<input type="radio"/>

7.1.5 History Data Output

M system	<input type="radio"/>
L system	<input type="radio"/>

7.1.7 System Configuration Data Output

M system	<input type="radio"/>
L system	<input type="radio"/>

7.2 Input/Output I/F

7.2.1 RS-232C I/F

M system	○
L system	○

There are 2 ports (port 1/2) available with the RS-232C interface for control unit.

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

Each port can be used for the following application.

Port 1: Input/Output, Tape operation

Port 2: Input/Output, Tape operation, GX Developer communication, computer link

7.2.2 IC Card I/F

7.2.2.2 Front IC Card I/F

M system	○
L system	○

Interface card to use a compact flash card can be attached in front of the NC control unit and used.

7.2.3 Ethernet I/F

M system	○
L system	○

Ethernet interface card can be attached onto the NC unit and used.

7.2.6 USB Memory I/F [Up to 2GByte]

M system	○
L system	○

A USB memory can be mounted.

Spindle, Tool and Miscellaneous Functions

8.1 Spindle Functions (S)

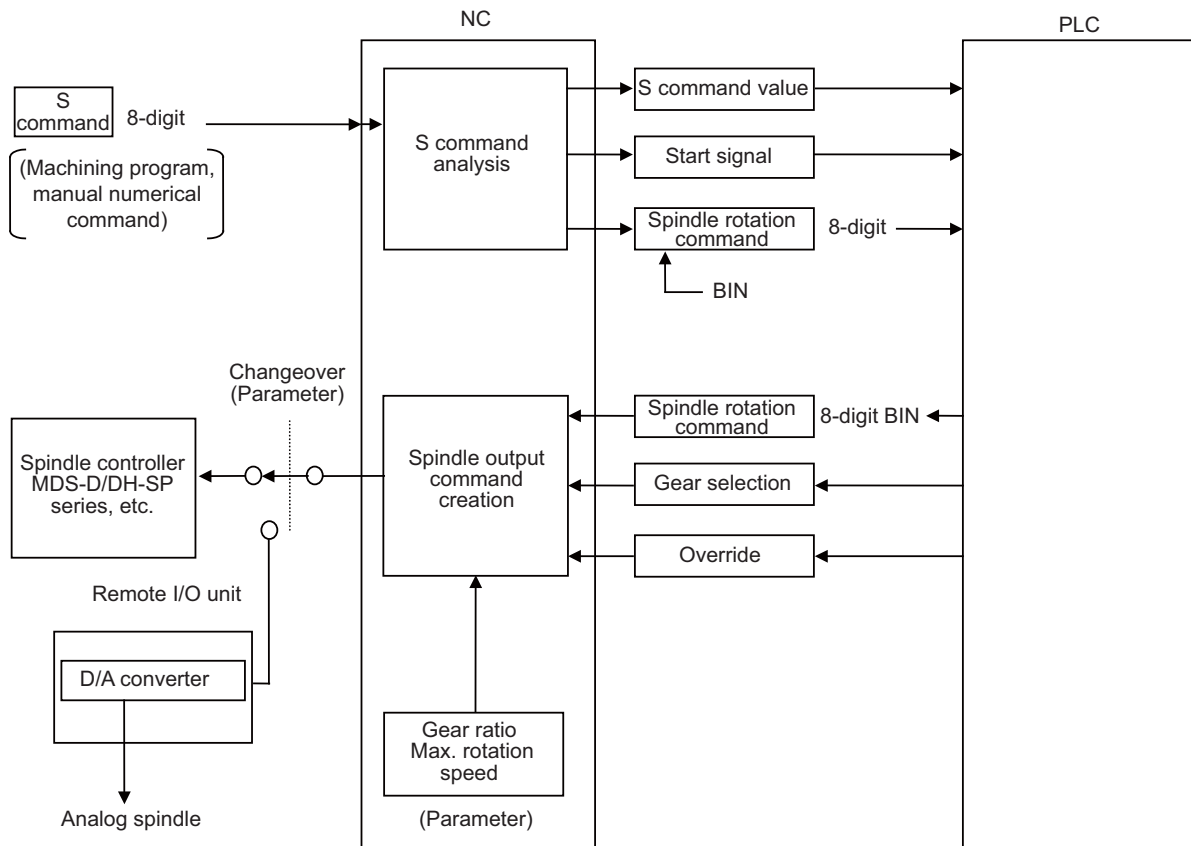
8.1.1 Spindle Control Functions

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.

Up to 1 set of S commands can be commanded in one block.

Processing and complete sequences must be incorporated on the PLC side for all S commands.



- (1) 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.
- (2) The number of gear steps can be commanded up to four steps.
- (3) The max. spindle rotation speed can be set for each gear.

8.1.1.1 Spindle Digital I/F

M system	<input type="radio"/>
L system	<input type="radio"/>

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

8.1.1.2 Spindle Analog I/F

M system	<input type="radio"/>
L system	<input type="radio"/>

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

8.1.1.3 Coil Switch

M system	<input type="radio"/>
L system	<input type="radio"/>

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

M system	<input type="radio"/>
L system	<input type="radio"/>

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.

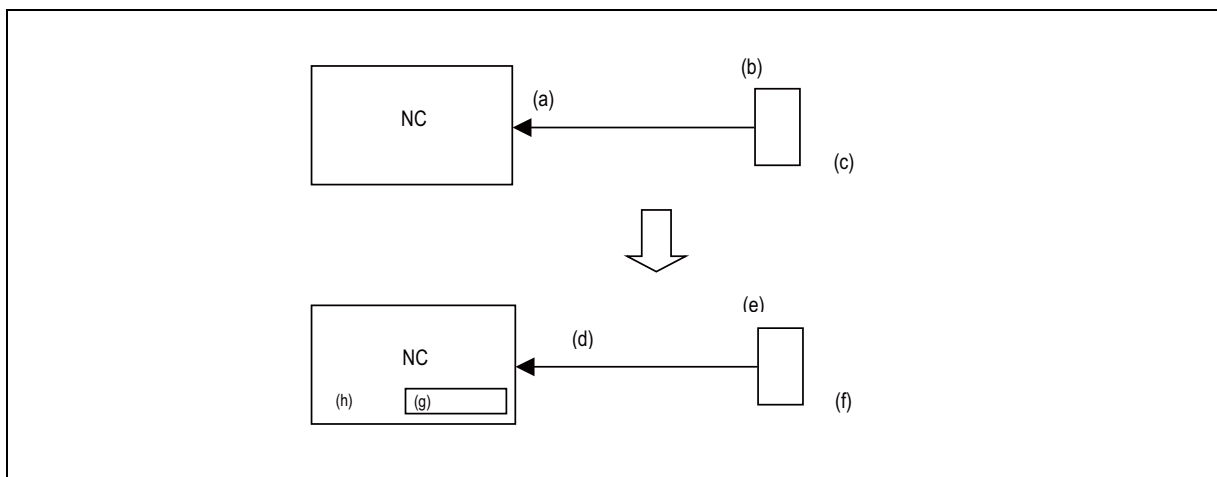
8.1.1.5 Encoder Input I/F

M system	○
L system	○

Encoder pulse input used to be fixed to 1024 pulse input on the conventional analogue interface. With this function, arbitrary pulse can be input by parameters set in R register.

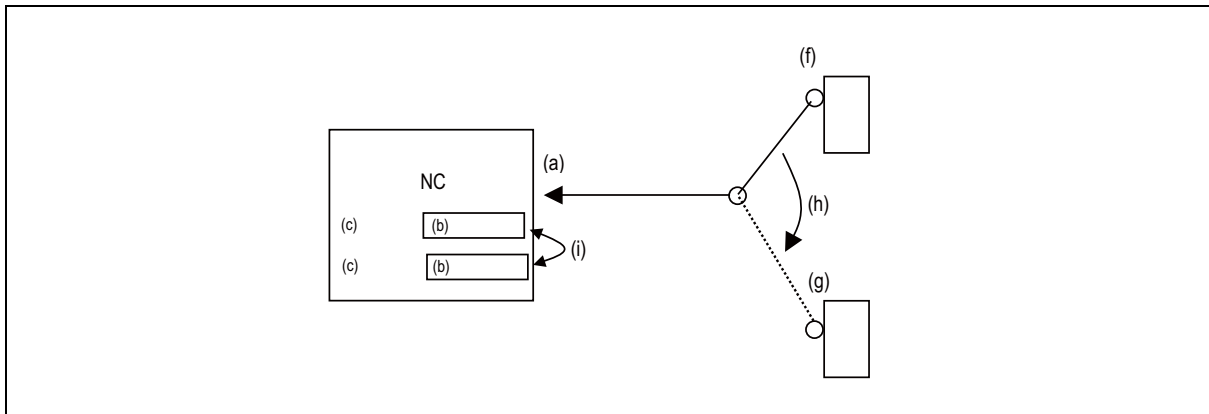
The maximum number of input pulse is 76800. Synchronous feed function can be activated with the arbitrary pulse if the number of pulse to be used is set in R register and the external signal is turned ON.

Encoder arbitrary pulse input



- (a) ENC interface
- (b) Encoder
- (c) 1024 pulse fixed
- (d) ENC interface
- (e) High resolution Encoder
- (f) Max76800 pulse
- (g) Input pulse
- (h) R register

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



- (a) ENC interface
- (b) Input pulse
- (c) R register
- (f) Encoder1 (High resolution)
- (g) Encoder2 (Low resolution)
- (h) Switch with ladder
- (i) Switch internally

8.1.2 S Code Output

M system	○
L system	○

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.

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.

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

8.1.3 Constant Surface Speed Control

M system	○
L system	○

With radial direction cutting, this function enables the spindle speed to be changed in accordance with changes in the radial direction coordinates and the workpiece to be cut with the cutting point always kept at a constant speed (constant surface speed).

G code	Function
G96	Constant surface speed
G97	Constant surface speed cancel

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.

(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.

8.1.4 Spindle Override

M system	○
L system	○

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

(1) Type 1 (code method)

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

(2) Type 2 (value setting method)

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

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

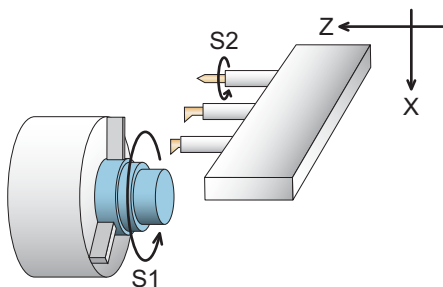
8.1.5 Multiple-spindle Control

Multiple-spindle control is a function that controls the second spindle for a machine tool equipped with another spindle (sub-spindle) in addition to the first spindle (main spindle).

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 (L system only)	Control based on an external 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 the second spindle.



8.1.5.1 Multiple-spindle Control I

M system	—
L system	○

(1) Spindle selection command

Using the spindle selection command (such as G43.1 [G group 20]), this function makes it possible to switch the spindle between the first and the second spindles to which the subsequent S command (S*****) is to apply.

Command format

G43.1 ; First spindle control mode ON
G44.1 ; Selected spindle control mode ON; the selected spindle number is set using a parameter.
G47.1 ; All spindles simultaneous control mode ON

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

In addition to using the "S*****" S commands, it is also possible to assign commands which differentiate the applicable spindle to either the first spindle or the second spindle by using the S =*****.

Command format

S O =***** ;	
O	:Number assigned as the spindle number (1: first spindle; 2: second spindle; 3: third spindle; 4: fourth spindle); variables can be designated.
*****	: Rotational speed or surface speed value assigned by 5-digit analog command; variables can be designated.

8.1.5.2 Multiple-spindle Control II

M system	—
L system	○

With this function, one S command is used to command to the spindle, and which spindle is selected is decided depending on a signal from the PLC.

A parameter is used to switch between multi-spindle control II and the conventional multi-spindle control I function.

Spindle command selection, spindle selection

The S command for the spindle is output as the rotation speed command to the spindle which has been selected by the spindle selection signal ON from the PLC. The selected spindle rotates at the rotation speed which was output.

The spindles which were de-selected by spindle selection signal OFF continue to rotate at the same rotation speed as the speed immediately before their de-selection. As a result, each of the spindles can be made to rotate simultaneously at a different rotation speed.

8.1.6 Spindle Orientation

M system	○
L system	○

This function stops the spindle rotation at a certain position.

When Z-phase has not yet been passed, pass Z-phase twice (excluding reciprocations within one rotation), and then position to the orientation position.

When Z-phase has been passed already, immediately position to the orientation position.

The spindle does not rotate if Z-phase has been passed and the orientation position is already established when orientation command is issued.

When the spindle has been rotating in the direction opposite from that of orientation rotation when the orientation command is issued, orientation operation will be executed after decelerating to stop.

(a) Orientation

This function stops the spindle rotation at a certain position when using the digital spindle.

When the orientation command is used, the spindle will rotate several times and then stop at the orientation point.

The orientation position differs depending on the detector.

- When the encoder orientation (PLG and external encoder/ring sensor) is used:
...At the Z-phase position
- When the magnetic sensor (proximity switch) is used:
...At the magnetic sensor installation position

(b) Multi-point orientation

This function performs orientation to a position other than the Z-phase position by inputting a shift amount with the parameter or PLC. The shift amount is 0 to 35999. (Unit: 360°/36000=0.01°)

(Note 1) Multi-point orientation cannot be executed when using the magnetic sensor.

(Note 2) Orientation is possible only when the gear ratio is 1:1 for the PLG orientation.

(The orientation is completed at the PLG encoder's Z-phase, so when using reduction gears, the orientation points will be generated at several points during one spindle rotation.)

(c) Orientation imposition advance output

This function turns the spindle in-position signal ON as soon as the spindle reaches within the second in-position width. 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 tact time.

(d) Proximity switch orientation

Proximity switch 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.

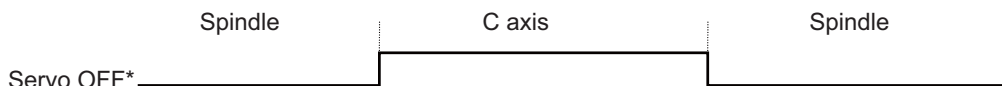
In the conventional spindle position zero point detection method, the spindle position zero point is calculated from the spindle drive unit and transmitted to NC when the position detector detects the one-time rotation signal (Z-phase) for the first time after the system has been turned ON, and the zero 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)

M system	○
L system	○

This function enables one spindle drive unit to be used also as the C axis (rotary axis) using an external signal. The C axis servo ON signal is used to switch between the spindle and C axis.



At servo OFF: ----- Spindle (C axis cannot be controlled).

At servo ON: ----- C axis (spindle cannot be controlled).

- Reference position return state

Reference position return is incomplete when the Z phase has not been passed.

Reference position return is complete when the Z phase has been passed.

- C axis position data

The NC's internal C axis position data is updated even for the spindle rotation during spindle control.

The C axis coordinate position counter is held during spindle control, and is updated according to the amount moved during spindle control when the C axis servo READY is turned ON. (The C axis position at servo ON may differ from the position just before the previous servo OFF.)

8.1.11 Spindle Speed Clamp

M system	○
L system	○

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 clamp speed 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).

8.3 Miscellaneous Functions (M)

8.3.1 Miscellaneous Functions

M system	○
L system	○

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.

This can be designated with an 8-digit number following address M (0 to 99999999). Up to four sets of M commands can be issued in one block. Note that the number M commands to be issued within the same block is determined by parameter.

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 ;

- (1) When 5 or more sets are commanded in one block, only the last 4 sets are valid.
M00, M01, M02, M30, M98 and M99 are used only for some specific purposes and cannot be assigned as regular M functions.
- (2) Processing and completion sequences are required for all M functions except 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.
- (3) 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.
- (4) 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.
 - (a) M function is executed after the movement is completed.
 - (b) 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 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

M system	○
L system	○

Up to four sets of M commands can be issued in a block.

Respective processing and completion sequences are required for all M commands included in a block (except 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

M system	○
L system	○

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.

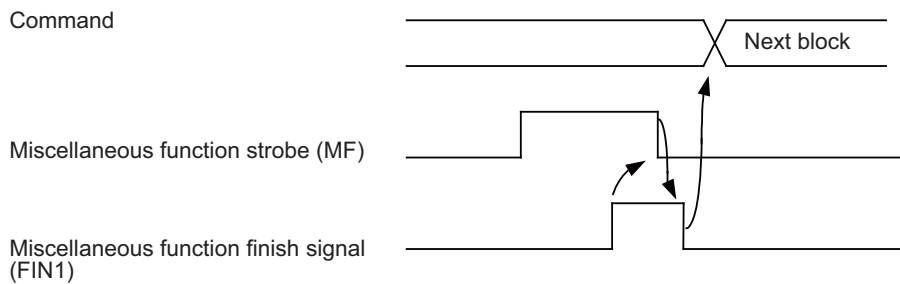
8.3.4 Miscellaneous Function Finish

M system	○
L system	○

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)

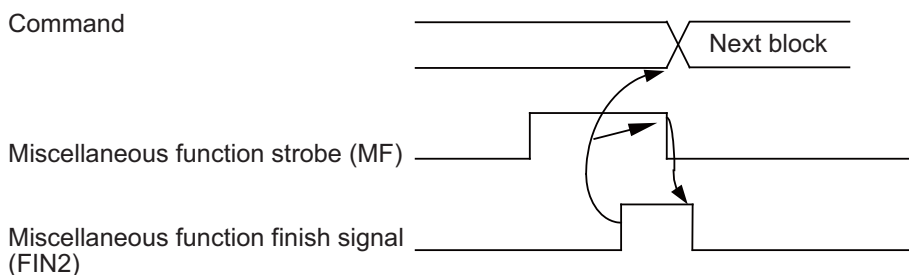
When the controller checks that FIN1 is ON, it sets the function strobes OFF. Furthermore, when the PLC checks that the function strobes are OFF, it sets FIN1 OFF. The controller checks that FIN1 is OFF and advances to the next block. Below is an example of a time chart applying when a miscellaneous 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.



8.3.6 Miscellaneous Command High-speed Output

M system	○
L system	○

The miscellaneous command high-speed output shortens a processing time per miscellaneous function. There are two functions that can be selected individually.

(1) 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 (MFIN1 to 4, SFIN1 to 6, TFIN1 to 4, BFIN1 to 4). 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.)

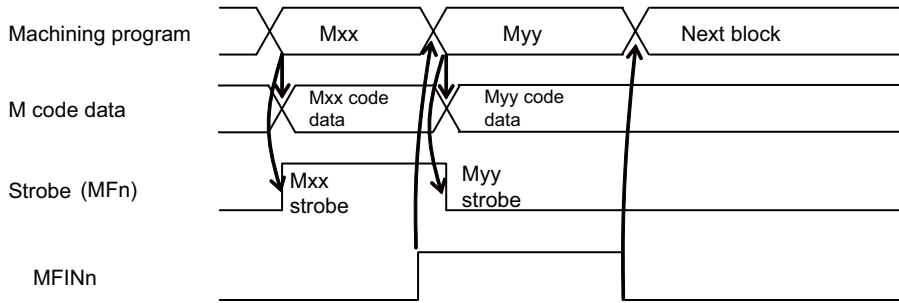
(2) Selecting miscellaneous command completion method

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.

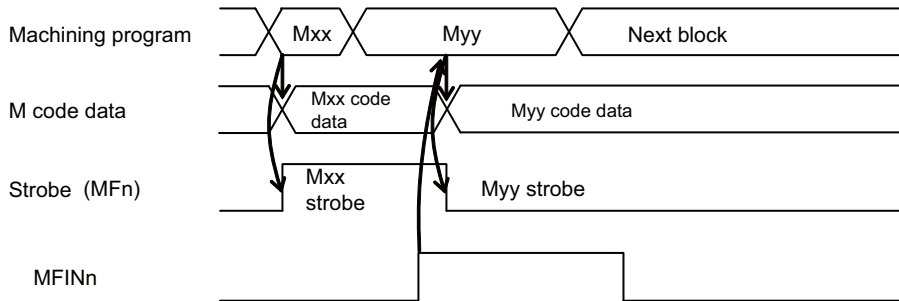
- High-speed method that waits for a finish signal from the PLC

The miscellaneous function completes by matching the logic level of the finish signal with that of the strobe signal.



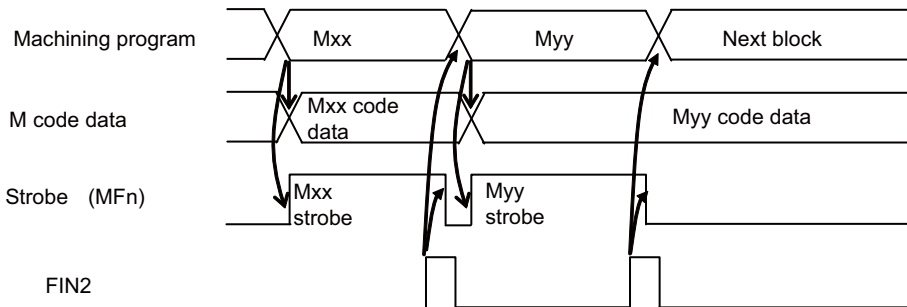
- High-speed method that does not wait for a finish signal from the PLC

When a series of miscellaneous commands is issued, the completion of the previous miscellaneous command is waited.



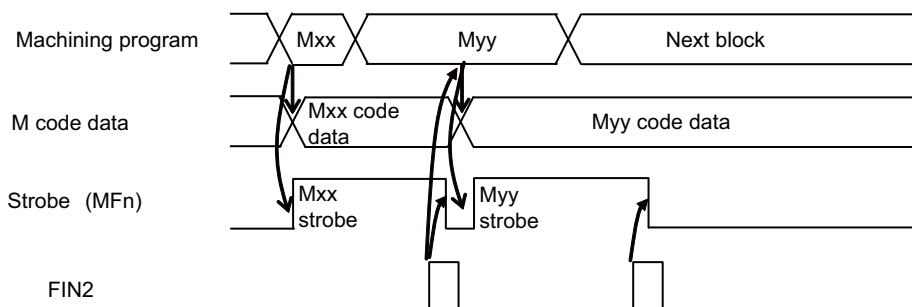
- Normal method that waits for a finish signal from the PLC

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

M system	<input type="radio"/>
L system	<input type="radio"/>

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.

(Note 2) There are some screens in the setting and display unit that cannot display all eight digits.

Tool Compensation

9.1 Tool Length/Tool Position

9.1.1 Tool Length Compensation

M system	○
L system	○

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.

(1) 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.

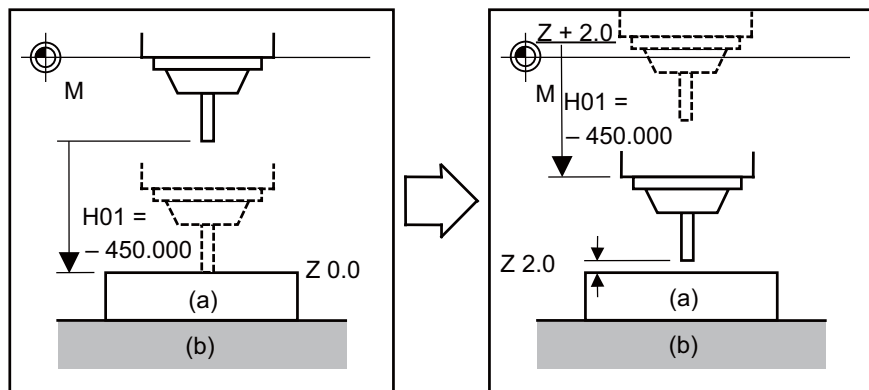
G49 ;
G43 H0 ;
G44 H0 ;

(Note) When the tool length compensation axis is returned to the reference position, the compensation of that axis is canceled.

(Example) Example of tool length compensation using a combination with tool length measurement type I

```
G28 X0 Y0 Z0
T01;
T02 M06;
G91 G00 G43
Z2.0 H01;
```

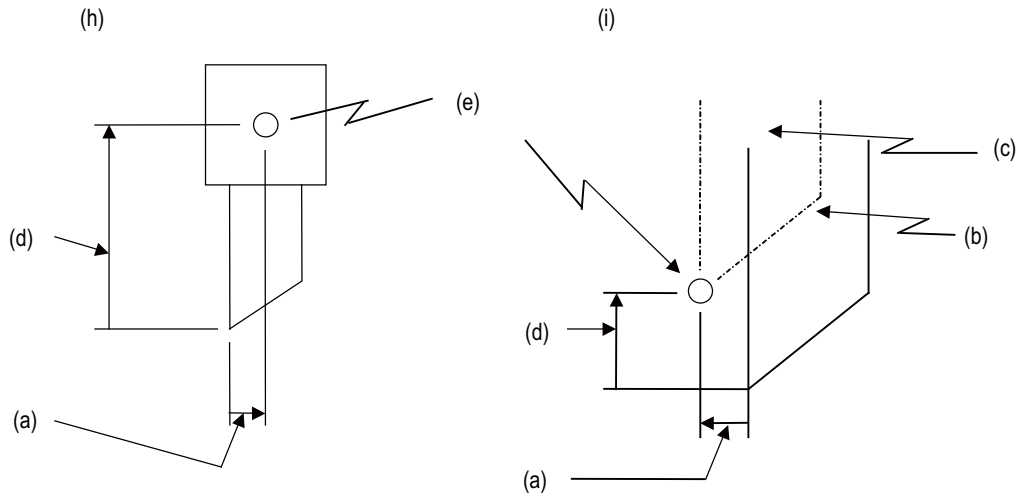
(Note) The tool length compensation amount is set as a negative value such as H01 = -450.000.



(a) Workpiece
(b) Table

(2) L system**(a) Shape compensation**

Tool length is compensated in reference to the programmed base position. The programmed base position is usually the center of the tool rest or the nose position of the base tool.



(a) Z-axis tool length compensation

(b) Tool used for machining

(c) Base tool

(d) X-axis tool length compensation

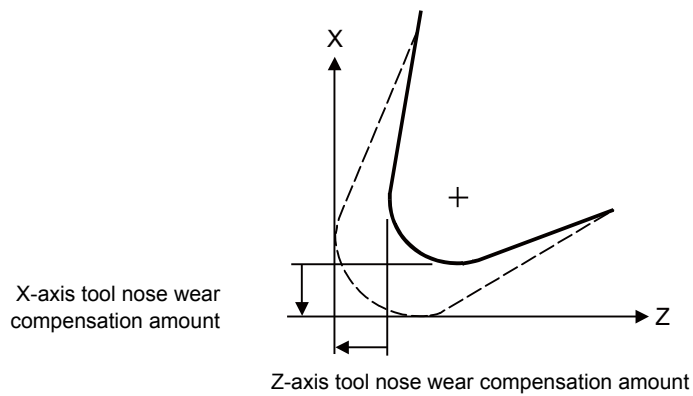
(e) Base position (base point)

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

(i) The programmed base position is the nose of the base tool:

(b) Wear compensation

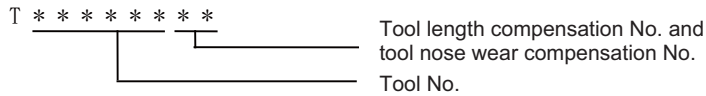
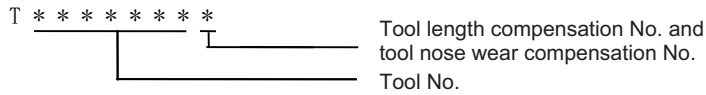
The wear of a tool nose can be compensated.



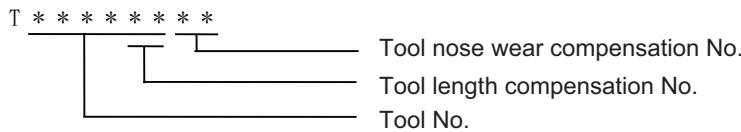
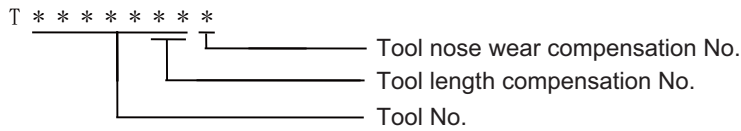
(c) Command format

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. The Nos. of such two types of compensations are specified by a parameter. Also a parameter is used to specify whether the compensation Nos. is specified by one or two low-order digits of a T command.

(i) Specifying tool length and wear compensation Nos. together using one or two low-order digits of the T command



(ii) 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) is added, the tool compensation will be validated for the additional axis.

The additional axis is the third or fourth axis which is selected using a parameter.

9.1.3 Tool Compensation for Additional Axes

M system	—
L system	○

The tool compensation for the lathe is valid for the X and Z axes. If an additional axis (Y axis) is added, the tool compensation will be validated for the additional axis.

9.2 Tool Radius

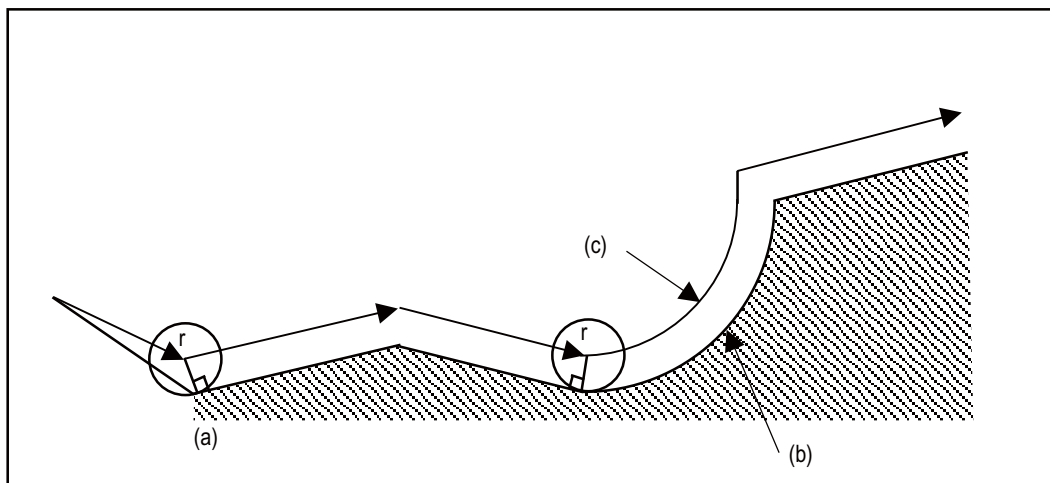
9.2.1 Tool Radius Compensation

M system	O
L system	—

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 cut 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



- (a) r: Tool radius compensation amount
- (b) Programmed path
- (c) Tool center path

The tool radius compensation command controls the compensation from that block in which G41 or G42 is commanded. In the tool radius 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.

```
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.

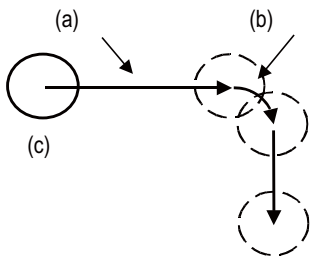
G17: XY plane, X, Y, I, J

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
```



(a) Tool center path
 (b) Arc inserted at corner
 (c) Programmed path

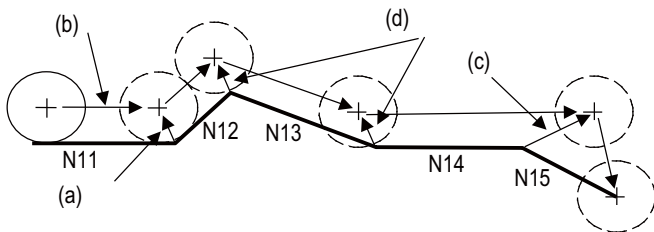
The compensation vector can be changed in following two ways.

```
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.



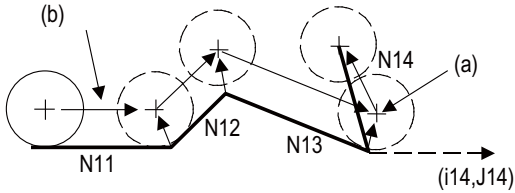
```
N11 G01 Xx11;
N12 G38 Xx12 Yy12;
N13 G38 Xx13 Yy13;
N14 G38 Xx14 Ii14 Jj14 Dd14;
N15 G40 Xx15 Yy15;
```

(a) Intersection point vector
 (b) Tool center path
 (c) Vector with length D (i14, j14)
 (d) Holding of previous intersection point vector

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.



```
N11 G01 Xx11 ;
N12 Xx12 Yy12 ;
N13 Xx13 Yy13 ;
N14 G40 Xx14 Ii14 Jj14 ;
```

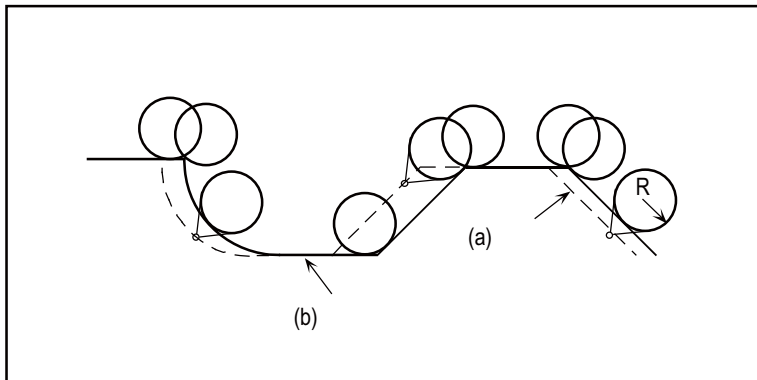
- (a) When i and j commands are assigned to G40
- (b) Tool center path

9.2.3 Tool Nose Radius Compensation (G40/41/42)

M system	—
L system	○

Corresponding to the tool No., the tool nose is assumed to be a half circle of radius R, and compensation is made so that the half circle touches the programmed path.

G code	Function
G40	Nose R compensation cancel
G41	Nose R compensation left command
G42	Nose R compensation right command



- (a) Compensated path
- (b) Programmed path

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.

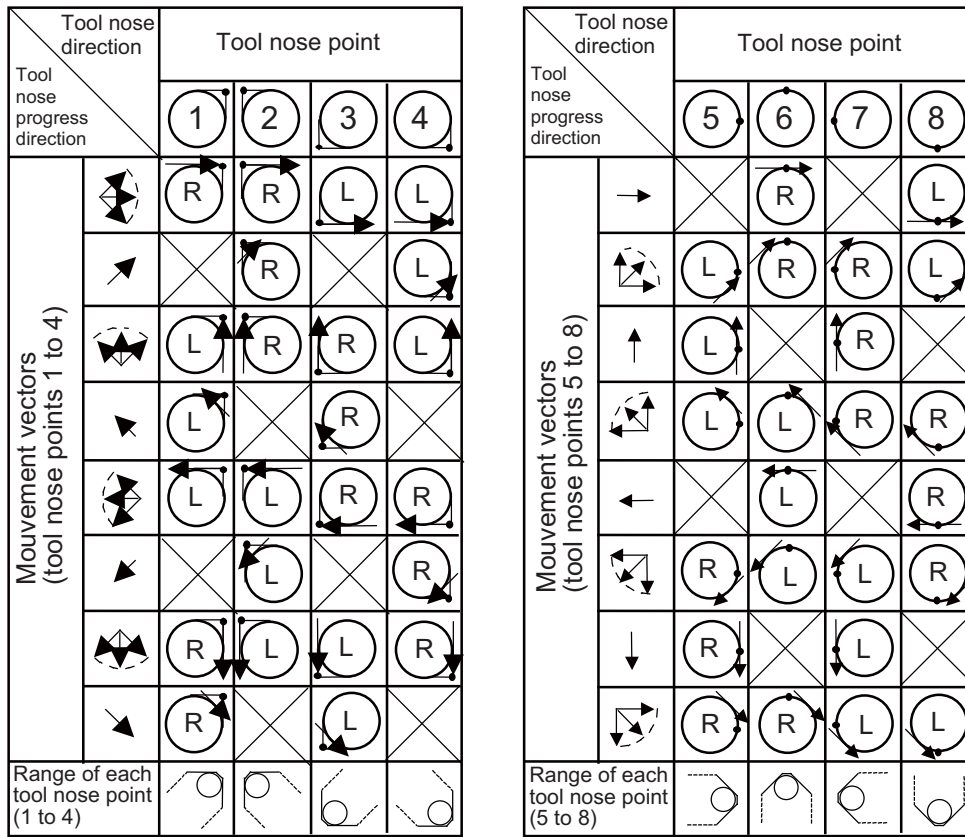
9.2.4 Automatic Decision of Nose Radius Compensation Direction (G46/40)

M system	—
L system	O

The nose radius compensation direction is automatically determined from the tool nose point and the specified movement vector.

G code	Function
G40	Nose radius compensation cancel
G46	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:



9.3 Tool Compensation Amount

9.3.1 Number of Tool Compensation Sets

9.3.1.3 80 Sets

M system	—
L system	O

9.3.1.4 200 Sets

M system	O
L system	—

9.3.2 Compensation Memory

9.3.2.1 Tool Shape/Wear Compensation Amount

M system	○
L system	○

This function registers the tool shape compensation and wear compensation amounts. Compensation may encompass two or more axes.

(1) 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.

(2) 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.

(a) 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.

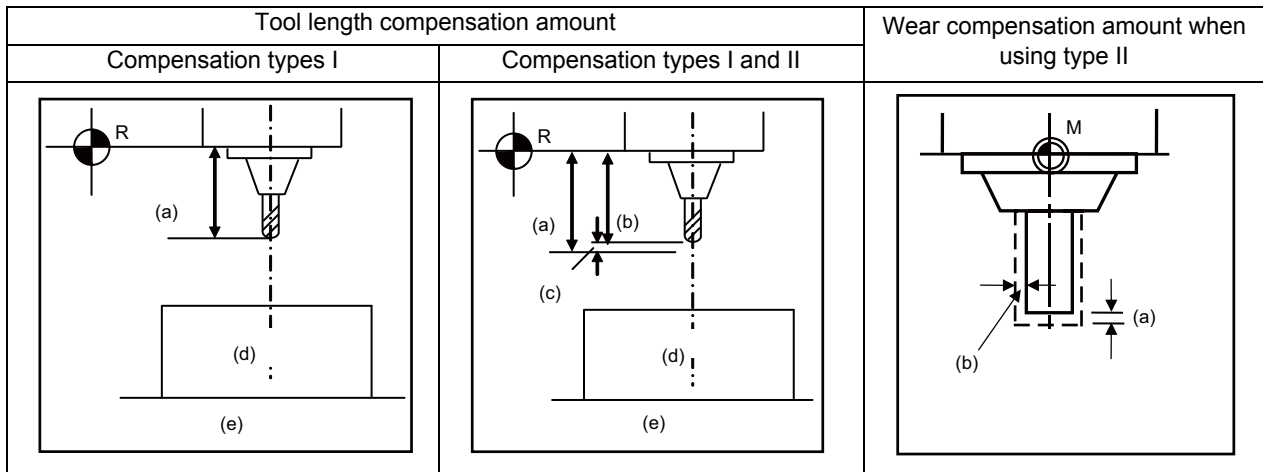
One compensation amount data is registered in one compensation No., and the compensation Nos. are assigned using the address D or H commands. When a No. is assigned by a D address command, compensation is provided in the form of the tool radius; when it is assigned by an H address command, it is provided in the form of the tool length.

(b) 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 compensated using the amount obtained by adding the shape compensation amount and tool length wear compensation amount.



- (a) Tool length compensation amount
- (b) Shape compensation amount
- (c) Tool radius wear compensation amount
- (d) Workpiece
- (e) Table

- (a) Tool length wear compensation amount
- (b) Tool radius wear compensation amount

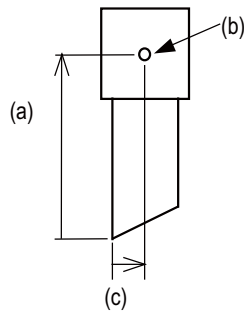
(c) Type III: 2-axis compensation amounts [L system]

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.

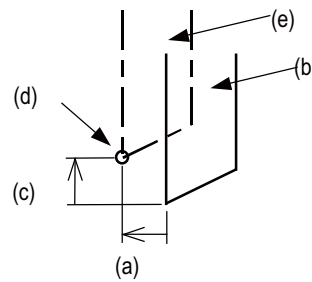
Compensation is carried out in the directions of the X, Y and Z axes from the base position in the program. Generally, the center of the tool rest or the tip of the base tool is used as the programmed base position.

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



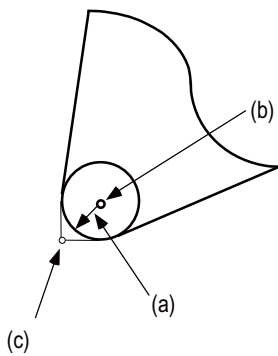
- (a) X-axis tool length compensation amount
- (b) Base position (base point)
- (c) Z-axis tool length compensation amount

2. The programmed base position is the tip of the base tool:

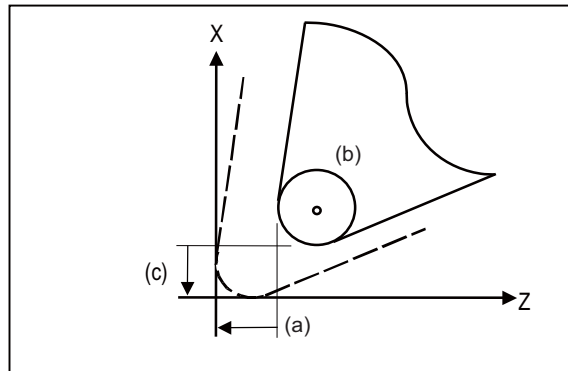


- (a) Z-axis tool length compensation amount
- (b) Tool used for machining
- (c) X-axis tool length compensation amount
- (d) Base position (base point)
- (e) Base tool

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.



- (a) Nose radius compensation amount
- (b) Tool nose center
- (c) Imaginary tool nose point



- (a) Z-axis tool length wear compensation amount
- (b) Tool nose
- (c) X-axis tool length wear 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.

The compensation type (I,II or III) is set using a parameter.

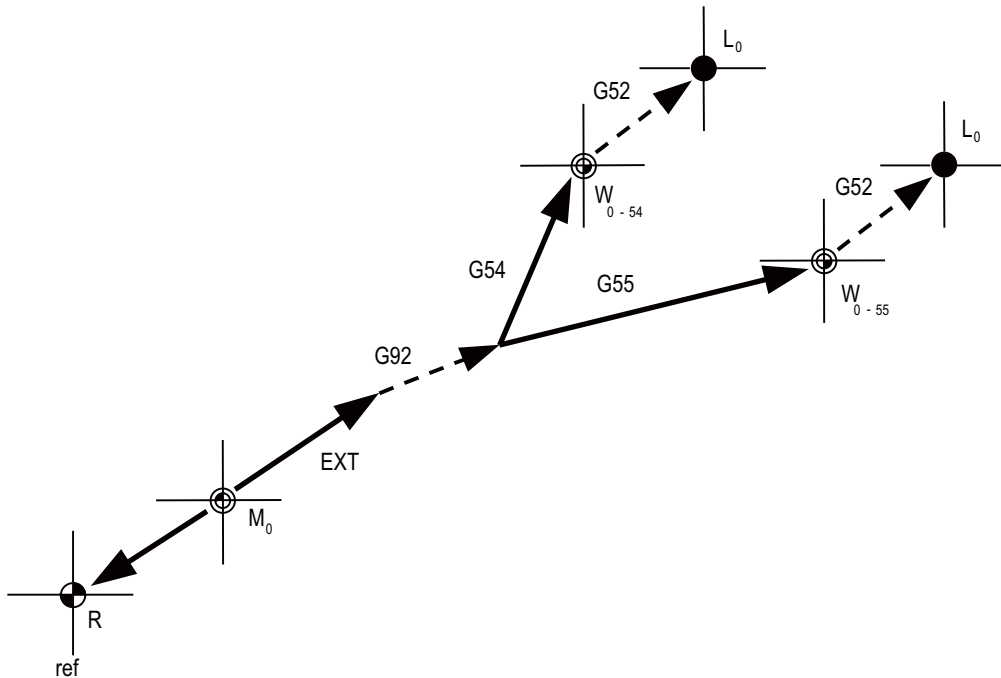
10

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
- G52 Local coordinate system offset (*1)
- W0-54 Workpiece coordinate system zero point (G54)
- W0-55 Workpiece coordinate system zero point (G55)
- G54 Workpiece coordinate system (G54) offset (*1)
- G55 Workpiece coordinate system (G55) offset
- G92 G92 coordinate system shift
- EXT External workpiece coordinate offset
- M0 Machine coordinate system zero point
- ref Reference position

- ▶ Offset set with parameters
- - -▶ Offset set with program
(0 when power is turned ON)

(*1) The G52 offset is available independently for G54 to G59.

10.1.1 Machine Coordinate System

M system	○
L system	○

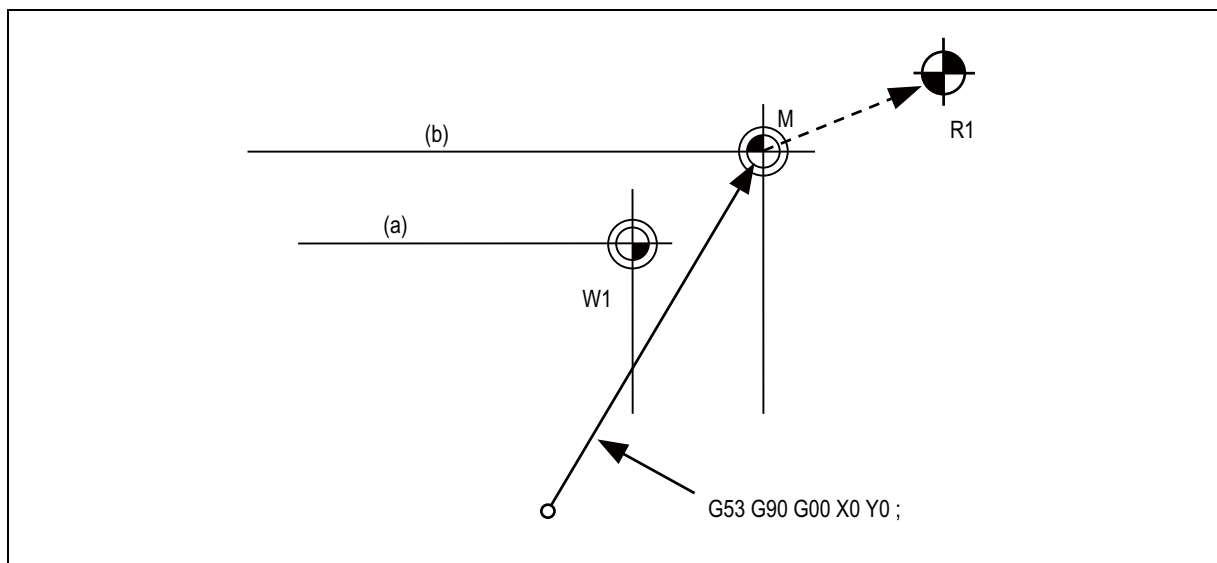
The machine coordinate system is used to express the prescribed positions (such as the tool change position and stroke end position) characteristic to the machine, and it is automatically set immediately upon completion of the first dog-type reference position return after the power has been turned ON or immediately after the power has been turned ON if the absolute position specifications apply.

The programming format for the commands to move the tool on the machine coordinate system is given below.

G53 (G90) (G00) Xx1 Yy1 Zz1 ;
 G53 : Coordinate system selection
 G90 : Incremental/absolute commands
 G00 : Movement mode [M system]
 Xx1,Yy1,Zz1 : End point coordinate on the machine coordinate system

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 unmodal command which is effective only in the block where it is assigned. The workpiece coordinate system being selected is not changed by this command.



(a) Workpiece coordinate system 1(G54)
 (b) Machine coordinate system(G53)
 R1: 1st reference position

10.1.2 Coordinate System Setting

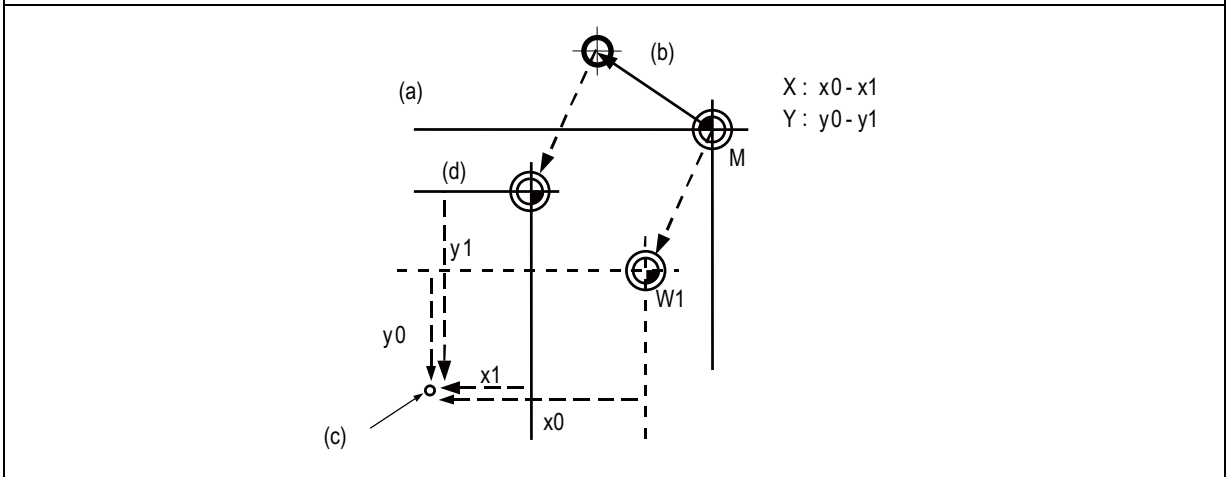
M system	<input type="radio"/>
L system	<input type="radio"/>

Among the workpiece coordinate systems with the G92, the program coordinate system (the programmed zero point) can be changed.

When a coordinate system setting is assigned using the G92 command, the G92 offset amount is applied so that the machine position in the current workpiece coordinate system is set to the coordinate position assigned by the G92 command, as shown in the figure below, and the workpiece coordinate systems are shifted accordingly. The machine does not run, and all the workpiece coordinate systems from G54 to G59 referenced to the machine coordinate system (or the external workpiece coordinate system if the external workpiece coordinate offset has been set) are shifted.

Offset of coordinate system by G92 coordinate system setting

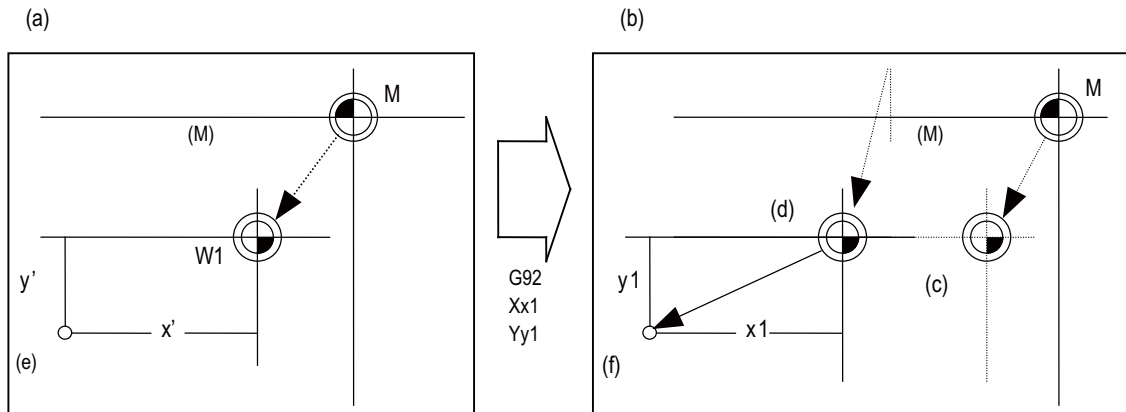
Example where W1 is shifted to new W1 when the machine was at the position (x0, y0) above W1 and the G92 Xx1 Yy1; command was assigned when the workpiece coordinate system W1 is modal (external workpiece coordinate system offset = 0; interrupt amount offset = 0)



- (a) Machine coordinate system
- (b) G92 offset amount
- (c) Machine position
- (d) New W1

The shifted coordinate system is returned to its original position by dog-type reference position return or the program.

When the coordinate system setting is commanded by G92, all the workpiece coordinate systems from G54 through G59 referenced to the machine coordinate system undergo a shift.



- (a) Coordinate system created by automatic coordinate system setting
- (b) Coordinate system after coordinate system setting by G92
- (c) Old W1
- (d) New W1
- (e) Tool position
- (f) G92 command position
- (M) Machine coordinate system

- (1) All the workpiece coordinates from G54 to G59 move in parallel.
- (2) There are two ways to return a shifted coordinate system to its original position.
 - (a) Carry out dog-type reference position return
 - (b) Move to machine coordinate system zero point and assign G92 and G53 commands in 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 from G54 to G59 to their original positions.

10.1.3 Automatic Coordinate System Setting

M system	○
L system	○

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.

- (1) Machine coordinate system corresponding to G53
- (2) G54 to G59 workpiece coordinate system
- (3) 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

10.1.4.1 Workpiece Coordinate System Selection (6 Sets)

M system	<input type="radio"/>
L system	<input type="radio"/>

When a multiple number of workpieces with the same shape are to be machined, these commands enable the same shape to be machined by executing a single machining program in the coordinate system of each workpiece.

Up to 6 workpiece coordinate systems can be selected.

The G54 workpiece coordinate system is selected when the power is turned ON or the reset signal which cancels the modal information is input.

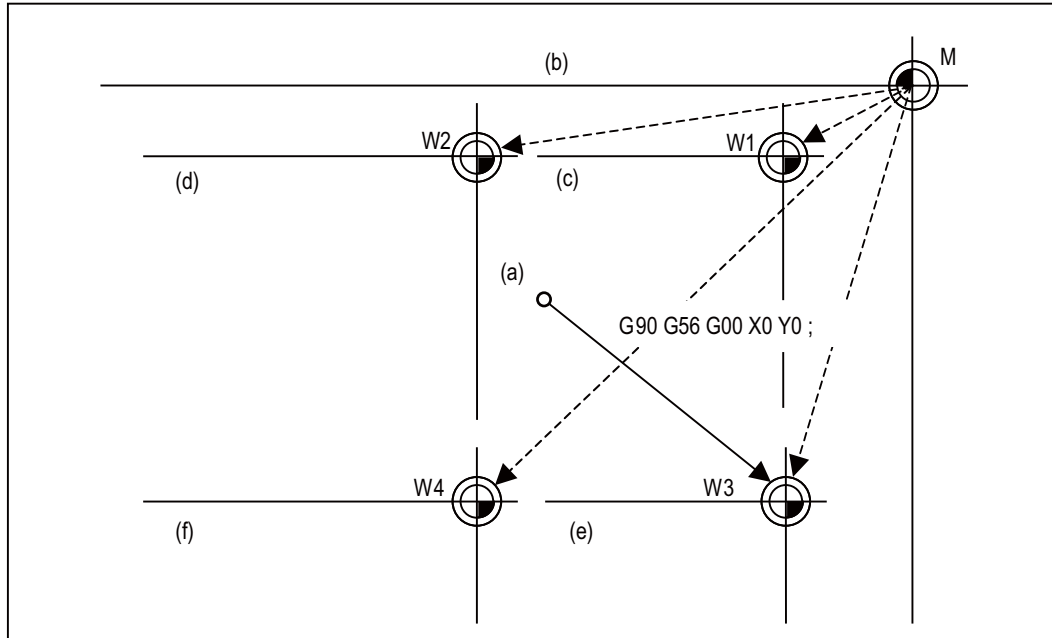
G code	Function
G54	Workpiece coordinate system 1 (W1)
G55	Workpiece coordinate system 2 (W2)
G56	Workpiece coordinate system 3 (W3)
G57	Workpiece coordinate system 4 (W4)
G58	Workpiece coordinate system 5 (W5)
G59	Workpiece coordinate system 6 (W6)

The command formats to select the workpiece coordinate system and to move on the workpiece coordinate system are given below.

(G90) G54 G00 Xx1 Yy1 Zz1 ;	
(G90)	: (Absolute command)
G54	: Coordinate system selection
G00	: Movement mode
Xx1,Yy1,Zz1	: Coordinate position of end point

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 three ways:

- (1) Setting using the setting and display unit
- (2) Setting using commands assigned from the machining program
- (3) Setting from the user PLC



- (a) Start
- (b) Machine coordinate system (G53)
- (c) Workpiece coordinate system 1 (G54)
- (d) Workpiece coordinate system 2 (G55)
- (e) Workpiece coordinate system 3 (G56)
- (f) Workpiece coordinate system 4 (G57)

10.1.4.2 Extended Workpiece Coordinate System Selection (48 Sets) G54.1P1 to P48

M system	○
L system	○

When a multiple number of workpieces with the same shape are to be machined, these commands enable the same shape to be machined by executing a single machining program in the coordinate system of each workpiece. 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.

```
(G90) G54.1Pn G00 Xx1 Yy1 Zz1 ;
G90                : (Absolute command)
G54.1Pn            : Coordinate system selection
G00                : Movement mode
Xx1,Yy1,Zz1       :Coordinate position of end point
```

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 three ways:

- (a) Setting using the setting and display unit
- (b) Setting using commands assigned from the machining program
- (c) Setting from the user PLC

(Note) While the G54.1Pn (extended workpiece coordinate system selection) is modal, the local coordinate offset is reduced to zero, and the G52 command cannot be used.

10.1.5 External Workpiece Coordinate Offset

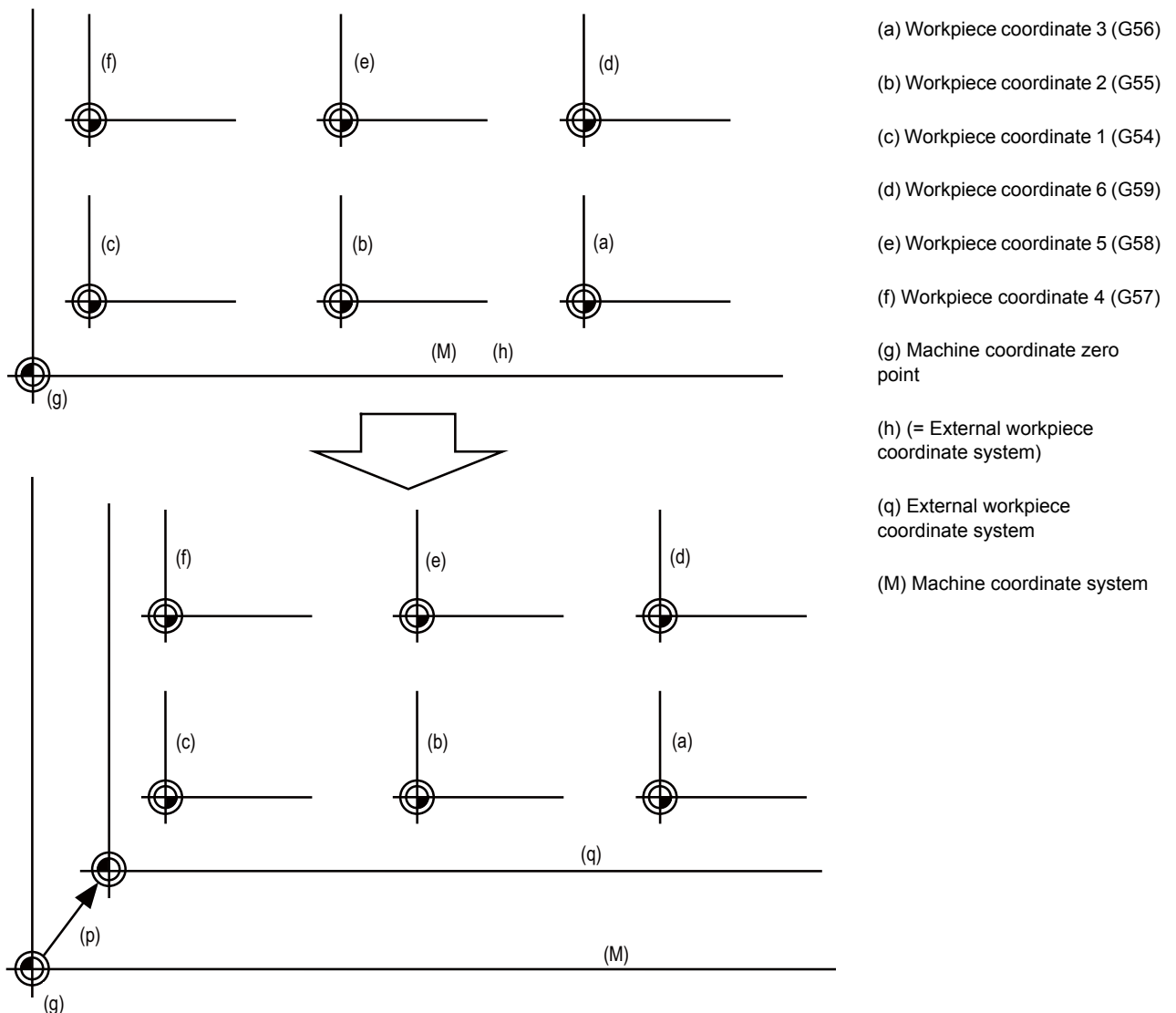
M system	○
L system	○

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)

M system	—
L system	○

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 ; (where α is an 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

M system	○
L system	○

This function is for assigning a coordinate system on the workpiece coordinate system now being selected. This enables the workpiece coordinate system to be changed temporarily.

The local coordinate system can be selected independently on each workpiece coordinate system G54 to G59.

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

The command format of the local coordinate system is given below.

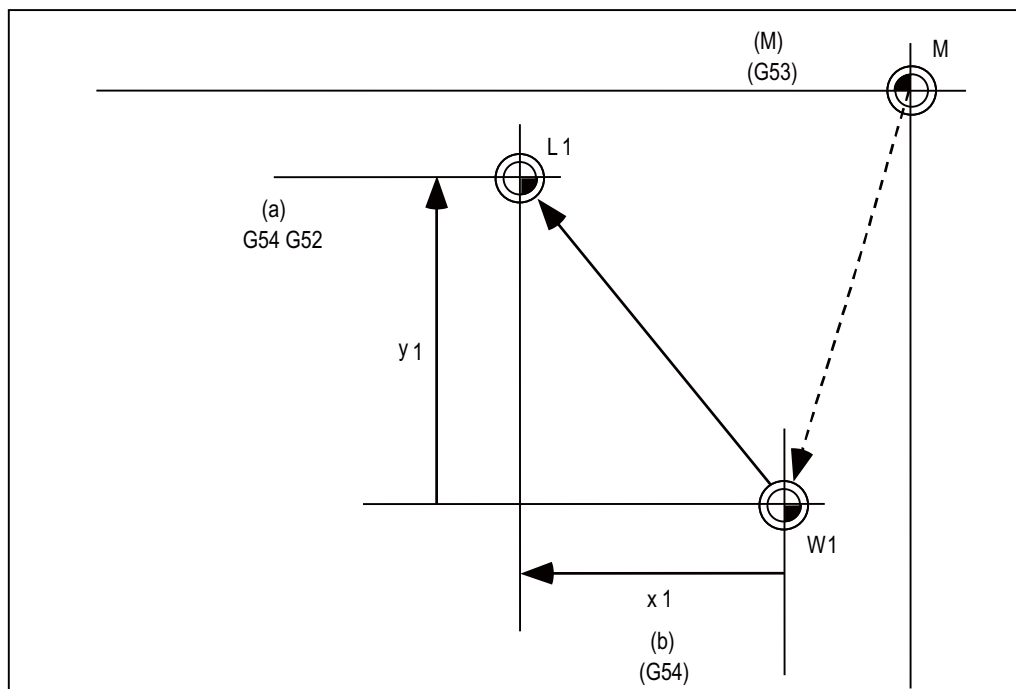
(G54) G52 Xx1 Yy1 Zz1 ;	
(G54)	: Workpiece coordinate system selection
G52	: Local coordinate system setting
Xx1,Yy1,Zz1	: Local coordinate offset amount

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 position setting 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 unmodal 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.



- (a) Local coordinate
- (M) Machine coordinate system
- (b) Workpiece coordinate 1

10.1.8 Coordinate System for Rotary Axis

M system	O
L system	O

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.

	Rotary axis				Linear axis
	Rotating type rotary axis		Linear type rotary axis		
	Short-cut invalid	Short-cut valid	Workpiece coordinate position linear type	All coordinate position linear type	
Workpiece coordinate position	Displayed in the range of 0° to 359.999°.		Displayed in the range of 0° to 99999.999°.		
Machine position/ relative position	Displayed in the range of 0° to 359.999°.		Displayed in the range of 0° to 99999.999°.		
ABS command	The incremental amount from the end point to the current position is divided by 360, and the axis moves by the remainder amount according to the sign.	Moves with a short-cut to the end point.	In the same manner as the normal linear axis, moves according to the sign by the amount obtained by subtracting the current position from the end point (without rounding up to 360 degrees.).		
INC command	Moves in the direction of the commanded sign by the commanded incremental amount starting at the current position.				
Reference position return	Follows the absolute/relative command for a movement to the interim position.				
	Returns to the reference position from the interim position within a 360 degree movement.		Moves and returns in the reference position direction for the difference from the current position to the reference position.		

10.1.9 Plane Selection

M system	○
L system	○

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

- (1) A parameter can be used to set either the X, Y or Z axis to which the additional axis is to be parallel.
- (2) 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.
- (3) The movement commands have no connection with the plane selection.

(Example)

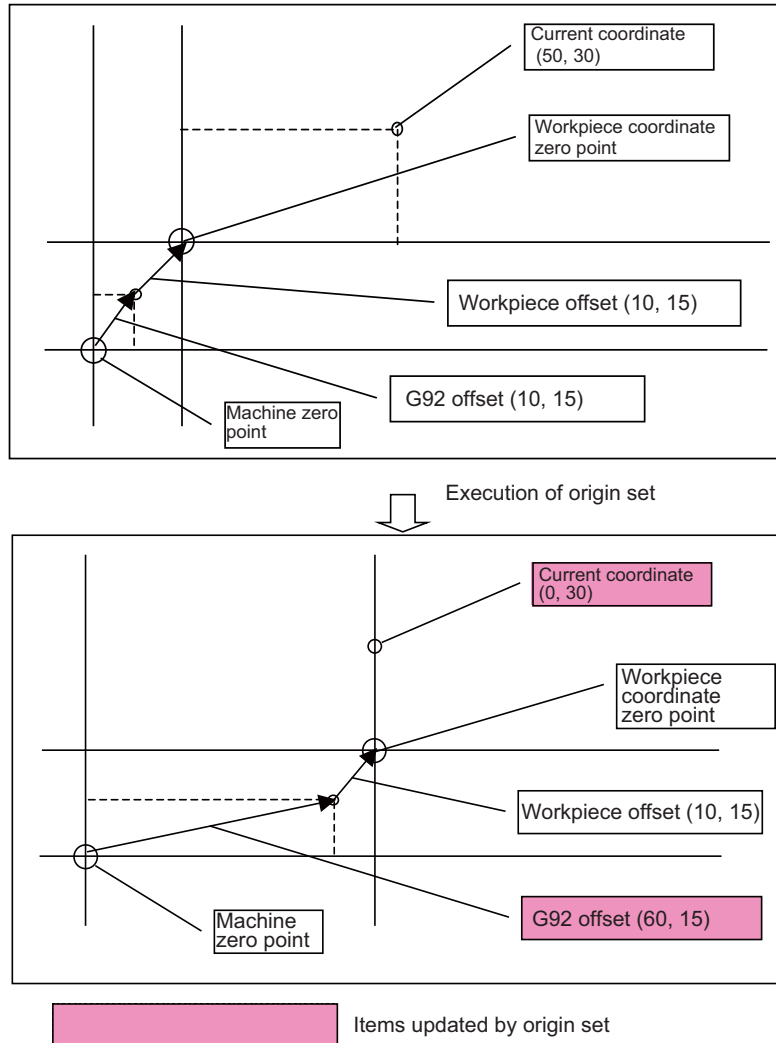
G19 X100. ;	With these program commands, X100. is the axis which does not exist on the G19 (Yp, Zp) plane, Yp-Zp plane is selected by G19 and the X axis moves by 100. mm separately from the plane selection.
G17 X100 . R50. ;	With these program commands, the Xp-Yp plane is selected by G17 and the arc command is controlled on the X-Y plane by this command.

10.1.10 Origin Set/Origin Cancel

M system	○
L system	○

(1) Origin set

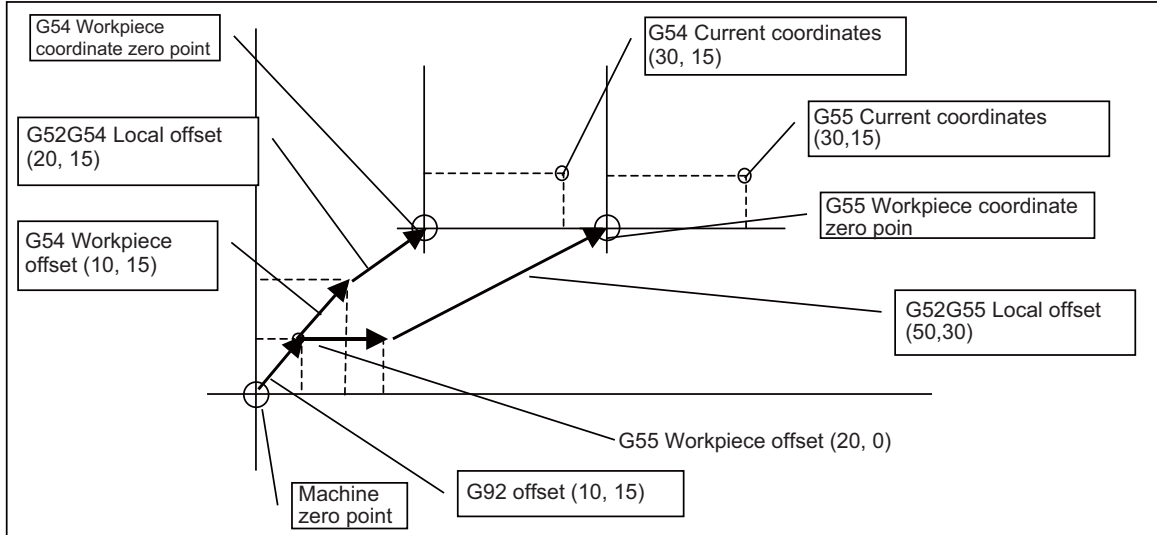
Origin set is a function that shifts the coordinate system so that the current position is the zero point on the workpiece coordinate system containing the workpiece coordinate system's offset value. The relative position counter and workpiece coordinate counter are set to "0" with this operation. In other words, this is the same as the coordinate system setting command "G92 X0;". (For target axis: X)



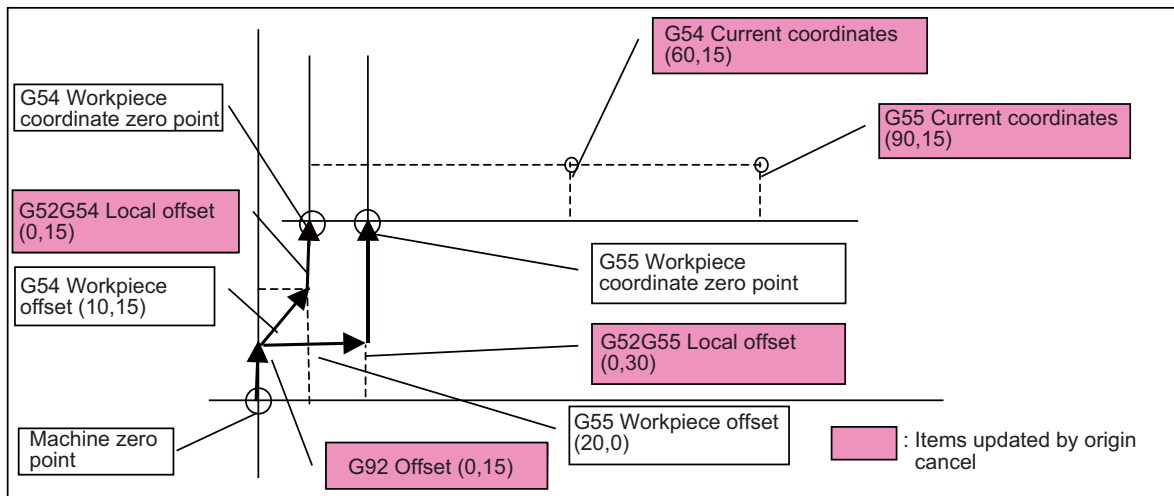
(2) Origin cancel

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

M system	<input type="radio"/>
L system	<input type="radio"/>

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 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.

10.2 Return

10.2.1 Manual Reference Position Return

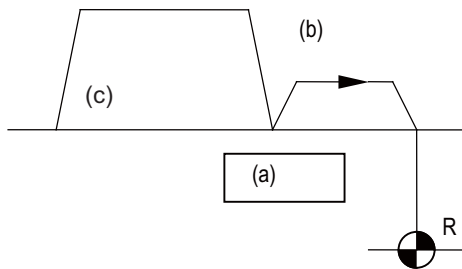
M system	○
L system	○

This function enables the tool to be returned manually to the position (reference position) which is characteristic to the machine.

(1) Return pattern to reference position

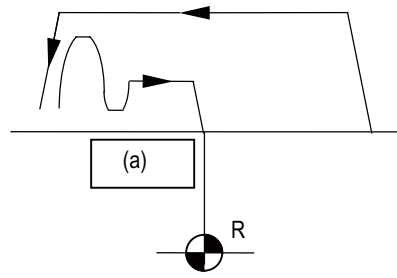
[Dog type]

When starting in same direction as final advance direction



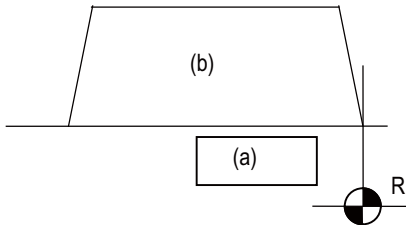
- (a) Dog
- (b) Creep speed
- (c) Reference position return speed

When starting in opposite direction as final advance direction



- (a) Dog

[High-speed type]



- (a) Dog
- (b) Rapid traverse rate

(2) 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

M system	○
L system	○

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.)

The G28 programming format is given below.

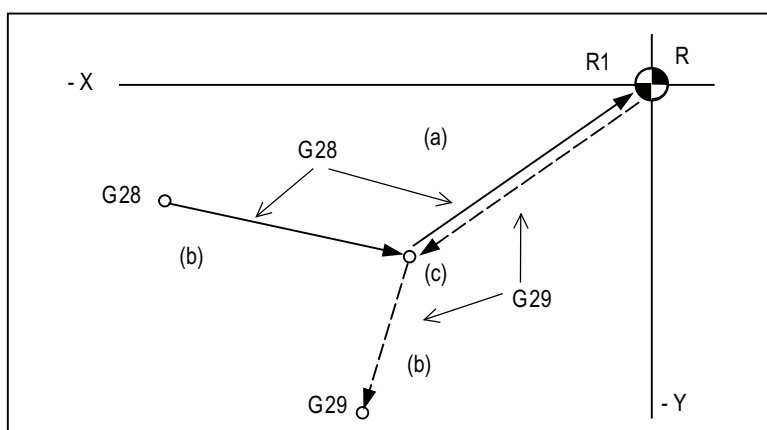
G28 Xx1 Yy1 Zz1 ;	
G28	: Return command
Xx1,Yy1,Zz1	: Return control axes (coordinate of interim point)

Tool is first positioned by rapid traverse to the assigned position (interim point) and then is returned to the 1st reference position for each axis independently.

The G29 programming format is given below.

G29 Xx1 Yy1 Zz1 ;	
G29	: Return command
Xx1,Yy1,Zz1	: Return control axes (coordinate of assigned position)

The tool is first moved by rapid traverse for each axis to the interim position which is passed through with G28 or G30, and is then positioned by rapid traverse at the position assigned by the program.



- (a) Non-interpolation movement
- (b) Interpolation or non-interpolation can be selected
- (c) Interim point
- R1 1st reference position

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.
- (Note 2) The number of axes for which reference position return can be performed simultaneously depends on the number of simultaneously controlled axes.
- (Note 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.
- (Note 4) If, at the time of the reference position return, the tool length compensation has not been canceled, it will be canceled and the compensation amount also cleared upon completion of reference position return. The tool length compensation can also be canceled temporarily using a parameter. In this case, however, the tool compensation is restored by the next movement command.
- (Note 5) 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.
- (Note 6) When a single block operation is selected, it can be chosen by parameter setting whether or not to enable interim point stop.

10.2.3 2nd, 3rd, 4th Reference Position Return

M system	○
L system	○

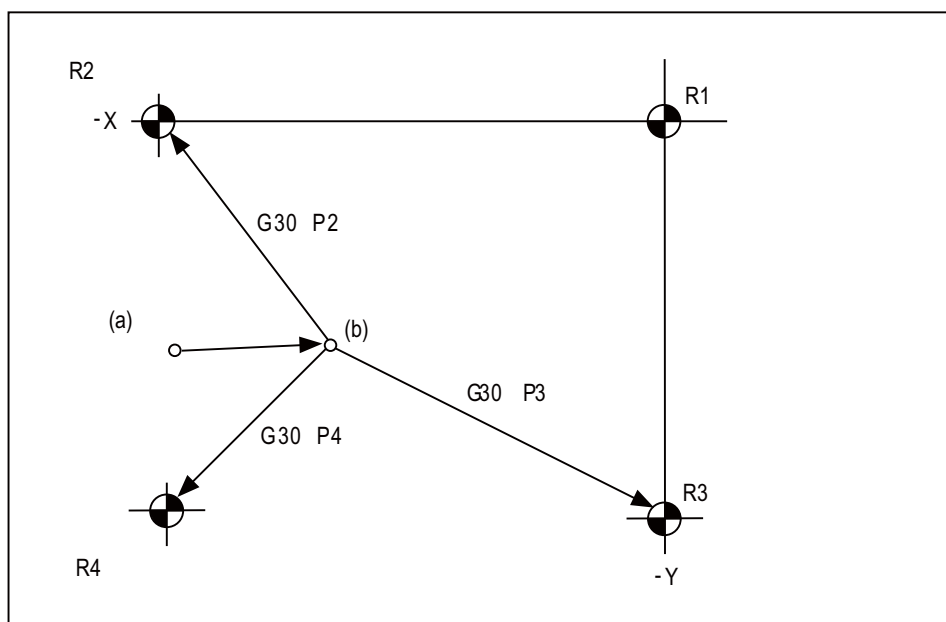
As with automatic 1st reference position return, commanding G30Pn during automatic operation enables the tool to be returned to the set points (2nd, 3rd or 4th reference positions) characteristic to the machine. The 2nd, 3rd and 4th reference positions can be set by parameters.

G code	Function
G30 P2	2nd reference position return
G30 P3	3rd reference position return
G30 P4	4th reference position return

The G30 programming format is given below.

G30 Xx1 Yy1 Zz1 Pp1 ;	
G30	: Return command
Xx1,Yy1,Zz1	: Return control axes (coordinate of interim point)
Pp1	: Return position No.

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.



- (a) Start point
- (b) Interim point
- 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.
- (Note 2) The number of axes for which reference position return can be performed simultaneously depends on the number of simultaneously controlled axes.
- (Note 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 command after the return.
- (Note 4) If, at the time of the reference position return, the tool length compensation has not been canceled, it will be canceled and the compensation amount also cleared upon completion of reference position return. The tool length compensation can also be canceled temporarily using a parameter. In this case, however, the tool compensation is restored by the next movement command.
- (Note 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.
- (Note 6) When a single block operation is selected, it can be chosen by parameter setting whether or not to enable interim point stop.

10.2.4 Reference Position Check

M system	○
L system	○

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 verification will be applied.

- (Note 1) The number of axes for which reference position check can be performed simultaneously depends on the number of simultaneously controlled axes.
- (Note 2) An alarm results unless the tool is positioned at the reference position upon completion of the command.
- (Note 3) Whether interpolation or non-interpolation is to apply to the movement can be selected using a parameter.

10.2.5 Absolute Position Detection

M system	○
L system	○

The absolute position detection function holds the relation of the actual machine position and the machine coordinates in the controller with a battery even when the power is turned OFF. When the power is turned ON again, automatic operation can be started without executing reference position return. (High-speed return will always be used for the reference position return command.)

For the absolute position detection method, there are two methods such as the dog-type and dog-less type according to how the zero point is established.

Method		Details	Establishment of zero point	Adjustment of zero point position
Dog-type		Same method as incremental detection dog-type	Zero point is established with dog-type reference position return completion.	The value is set in the parameter of zero point shift amount.
Dog-less type	Marked point method	The zero point position is set from the screen.	The zero point is established by input from the zero point initialization screen.	The value equivalent to the shift amount is set in the zero point initialization screen.
	Machine 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.	The value equivalent to the shift amount is set in the zero point initialization screen.

(Note) This function is valid for the NC axis and the PLC axis. This function cannot be used for the spindle and the auxiliary axis.

Diagnosis during absolute position detection

- (1) The machine position at power OFF and ON can be confirmed on the absolute position monitor screen.
- (2) If the amount that the axis is moved during power OFF exceeds the tolerable value (parameter), a warning signal will be output.
- (3) An alarm will be output if the absolute position information is lost.
- (4) An alarm will be output if the voltage of the battery for backing up the absolute position data drops.

10.2.6 Tool Exchange Position Return

M system	O
L system	O

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.

(Note 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.

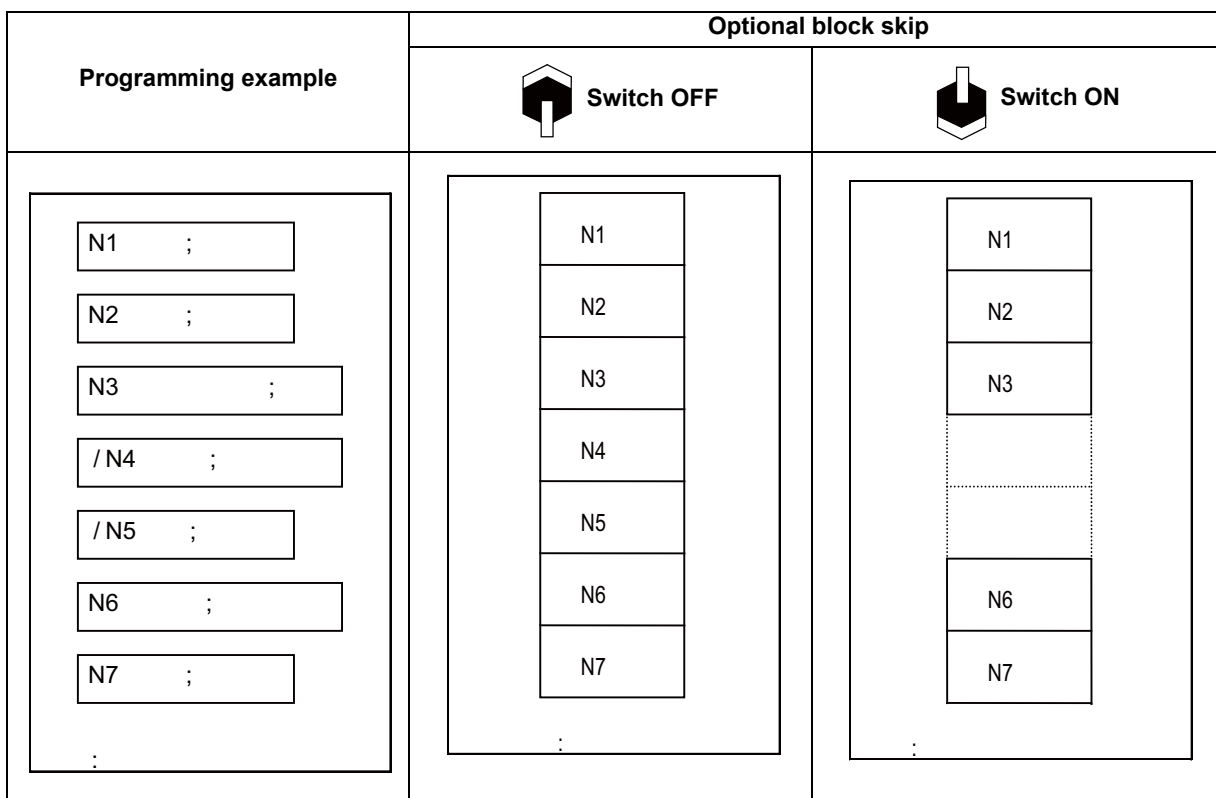
Operation Support Functions

11.1 Program Control

11.1.1 Optional Block Skip

M system	<input type="radio"/>
L system	<input type="radio"/>

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.



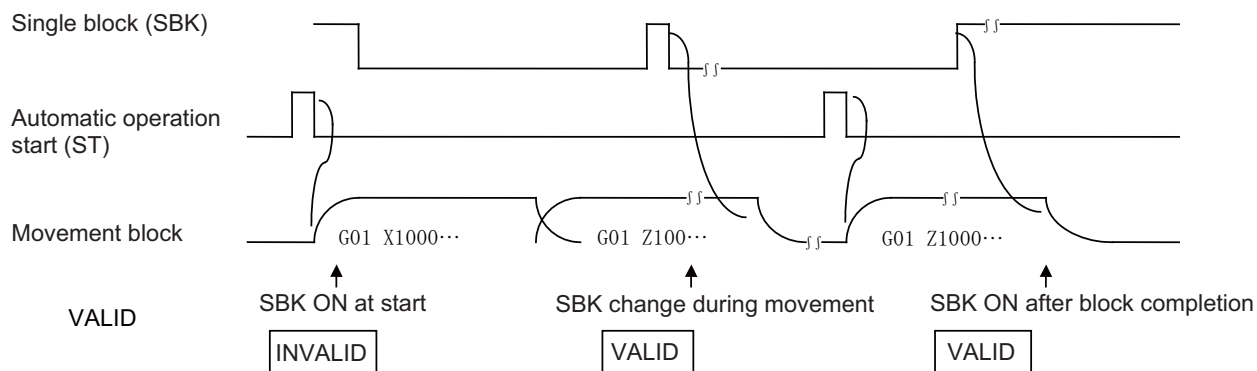
Skipping from a "/" in the middle of a program to the end of the program can also be enabled by parameter setting.

11.1.3 Single Block

M system	○
L system	○

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.



11.2 Program Test

11.2.1 Dry Run

M system	○
L system	○

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 feed rate	Rapid traverse rate
G01,G02,G03	Manual feed rate	Cutting clamp speed

11.2.2 Machine Lock

M system	○
L system	○

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.

- (1) 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.
- (2) Machine lock is effective in the signal status applying when the axis has stopped.
- (3) 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

M system	○
L system	○

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.

- (1) 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.
- (2) 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).
- (3) 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.
- (4) 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

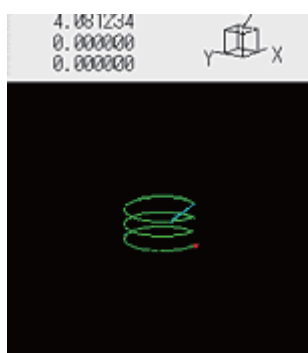
11.2.4.1 Graphic Check

M system	<input type="radio"/>
L system	<input type="radio"/>

This function traces the machining program movement path without executing automatic operation, and draws results of the NC internal operation. The machining program can be checked while drawing the figure accurately at a high speed. The drawing's viewpoint can be moved, enlarged and reduced while drawing. The three axes displayed are set with the parameters.

(Note 1) This function is required when performing program check for 2D.

(Note 2) Automatic operation is disabled during graphic check.



11.2.5 Graphic Trace

11.2.5.1 Graphic Trace

M system	<input type="radio"/>
L system	<input type="radio"/>

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, and 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.

11.2.6 Machining Time Computation

M system	<input type="radio"/>
L system	<input type="radio"/>

This function analyzes the machining program without moving the axis and calculates the approximate time required for machining.

The graphic check option is required to calculate the machining time.

The machining time is displayed as "hour: minute: second".

The miscellaneous function (M), spindle function (S), tool function (T) and 2nd miscellaneous function execution times differ according to the sequence and each function, and are not included in the calculated machining time.

11.3 Program Search/Start/Stop

11.3.1 Program Search

M system	○
L system	○

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

M system	○
L system	○

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.

11.3.4 Program Restart

M system	○
L system	○

When a machining program is to be resumed after it has been suspended midway due to tool damage or for some other reason, this function searches the program and the block to be resumed and enables machining to be resumed from the block.

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

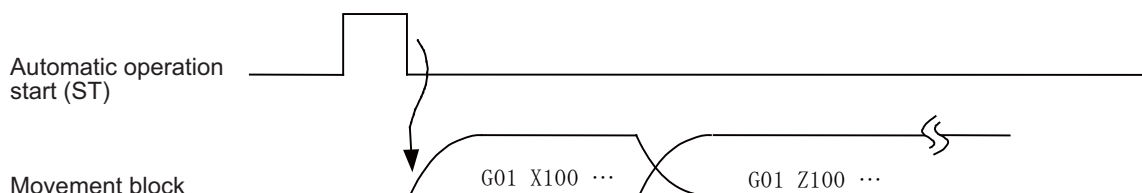
Restart method	Details	
Restart type 1	After machining is reset due to a tool breakage, etc., machining is restarted from the designated sequence number and/or block number. Only the program which had been executed just before can be restarted. Even after the power is turned ON again, the program can be restarted if the program has been executed before the power supply is turned OFF.	
Restart type 2	Automatic top search OFF	A top search must be executed from the screen. Then, command a sequence No. and block No., and restart a program.
	Automatic top search ON	A top search may not be executed from the screen. A top search is automatically executed, so it is not necessary to execute a top search from the screen. Command a sequence No. and block No., and restart a machining. An arbitrary program can be restarted by designating the program No. When a program No. is omitted, the program currently searched is restarted.

The validity of the automatic top search can be changed with control parameter.

11.3.5 Automatic Operation Start

M system	○
L system	○

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).



11.3.6 NC Reset

M system	○
L system	○

This function enables the controller to be reset.

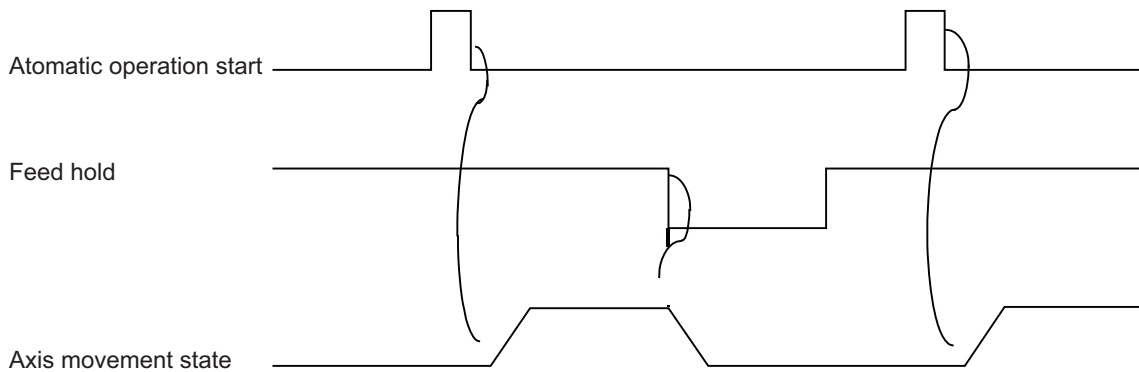
		Reset 1	Reset 2	Reset & Rewind
1	G command modals	Retained	Initialized	Initialized
2	Tool compensation data	Retained	Canceled (no operations)	Canceled
3	Memory indexing	Executed	Not executed	Executed
4	Errors/alarms	Reset	Reset	Reset
5	M, S and T code outputs	Retained	Retained	Retained
6	M code independent output	OFF	OFF	OFF
7	Control axis moving	Decelerated and stopped	Decelerated and stopped	Decelerated and stopped
8	Output signals	"In reset" signal	"In reset" signal	"In reset" signal "In rewind" signal

11.3.7 Feed Hold

M system	○
L system	○

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.

- (1) When the feed hold mode is entered during automatic start, the machine feed is stopped immediately, but the M, S, T and B commands in the same block are still executed as programmed.
- (2) 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.
- (3) 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

M system	○
L system	○

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.

11.4 Interrupt Operation

11.4.1 Manual Interruption

M system	○
L system	○

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 mode to manual mode
- Initiates the interrupt by selecting the manual mode at the same time as the automatic 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

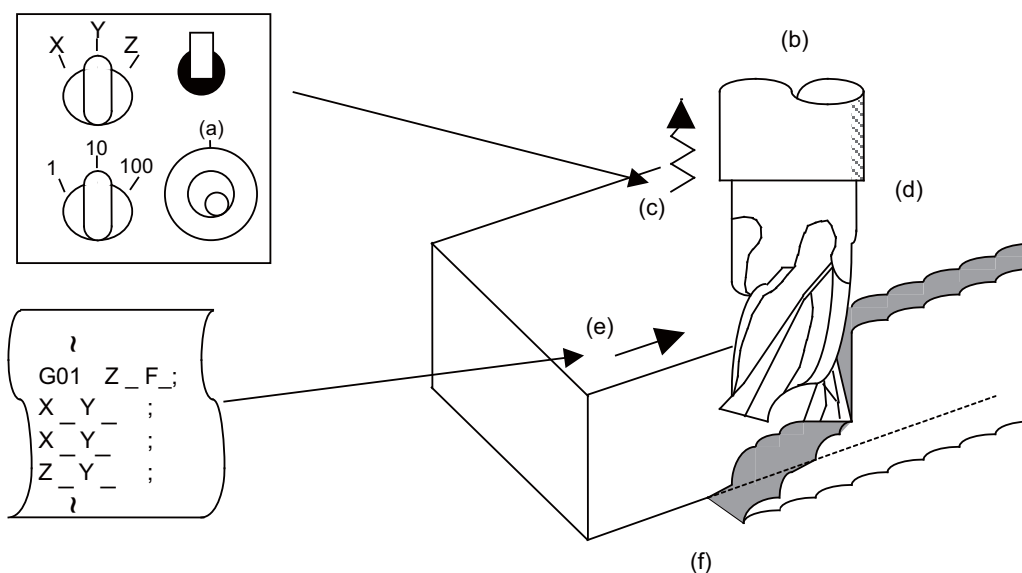
M system	○
L system	○

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.



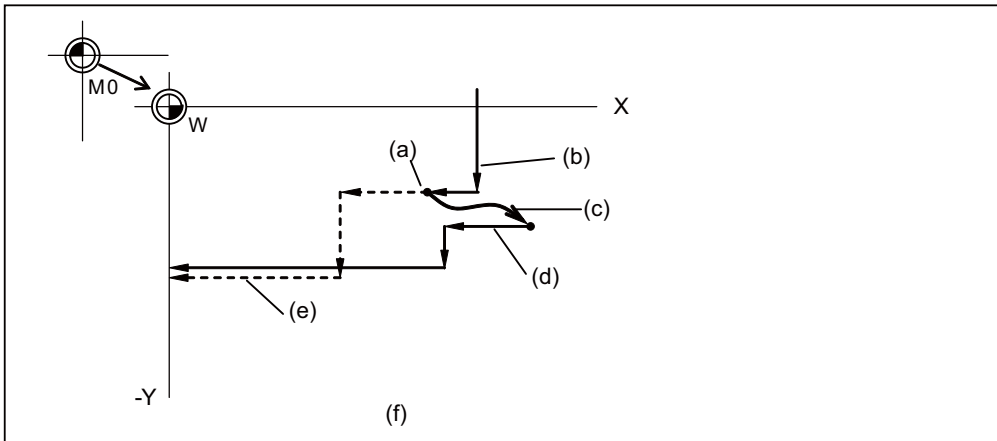
- (a) Interrupt
- (b) Tool
- (c) Handle feed
- (d) Workpiece
- (e) Automatic feed
- (f) Feed path with automatic feed and handle feed superimposed

11.4.3 Manual Absolute Switch

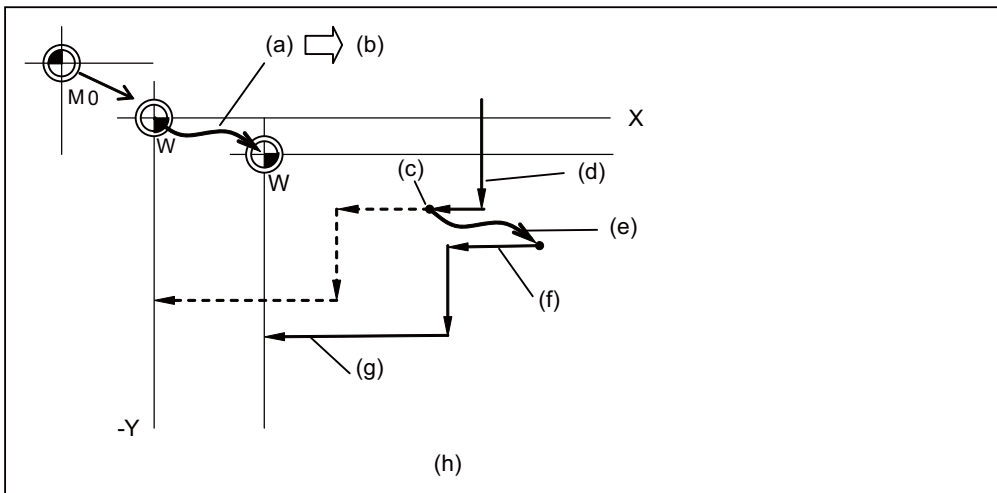
M system	<input type="radio"/>
L system	<input type="radio"/>

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.



- (a) Feed hold stop
- (b) Programmed path (absolute command)
- (c) Manual interrupt (The coordinate system W is not shifted)
- (d) Path after manual interrupt
- (e) Tool passes along same path as that programmed
- (f) With manual absolute signal ON



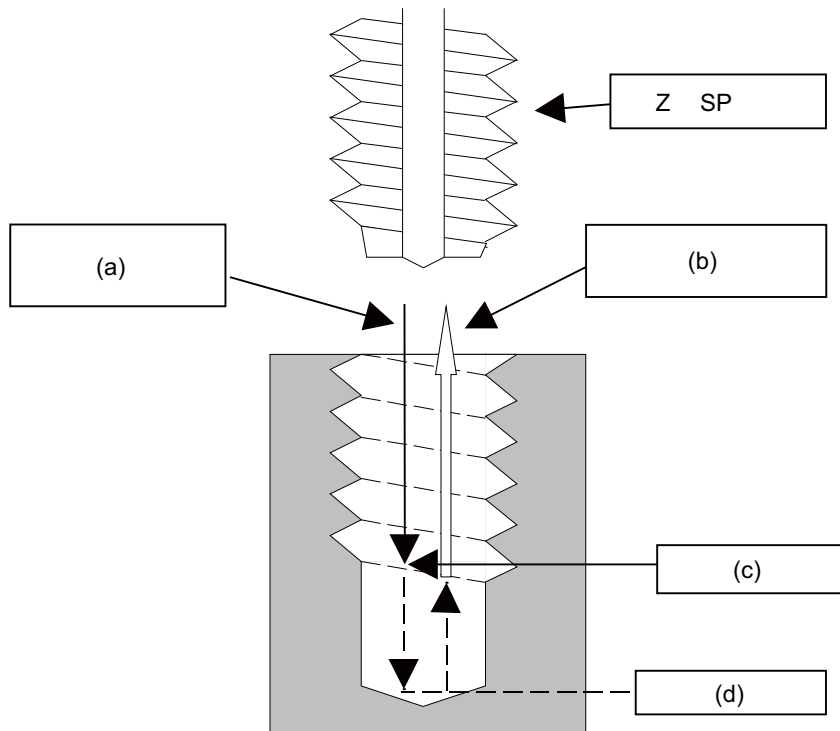
- (a) Manual interrupt (The coordinate system W is shifted)
- (b) Manual interrupt amount
- (c) Feed hold stop
- (d) Programmed path (absolute command)
- (e) Manual interrupt (The coordinate system W is shifted)
- (f) Path after manual interrupt
- (g) Path is shifted by an amount equivalent to manual interrupt value (Zero point of the coordinate system W moves)
- (h) With manual absolute signal OFF

The manual absolute ON state will be entered when the power is turned ON.

11.4.5 Tapping Retract

M system	<input type="radio"/>
L system	<input type="radio"/>

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.



- (a) Tap feed (spindle forward)
- SP (spindle)
- (b) Tap retract (spindle reverse)
- (c) Retract signal
- (d) Tap bottom

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.

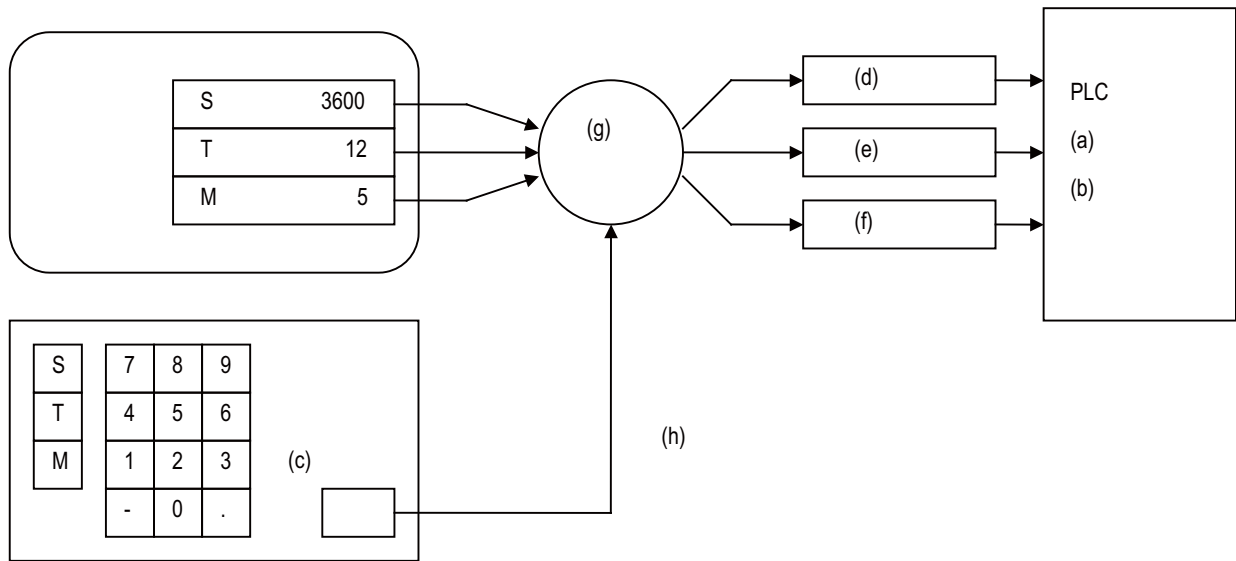
11.4.6 Manual Numerical Value Command

M system	<input type="radio"/>
L system	<input type="radio"/>

On the screen of the setting and display unit, the M, S and T (and B when 2nd miscellaneous function is valid) commands can be executed by setting numerical values and pressing [INPUT].

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. Even in an automatic operation mode, these operations can be conducted with block stop.

Furthermore, the M and T commands can be issued even on the tool compensation amount setting and display screen, therefore at the manual tool length measurement, the tools can be called successively to the spindle and measured very simply without having to change the screen page.



- (a) sequence
- (b) processing
- (c) Input
- (d) S command value
- (e) T command value
- (f) M command value
- (g) Manual numerical value
- (h) (Note) The input operation starts the execution of the M, S or T command.

11.4.8 MDI Interruption

M system	<input type="radio"/>
L system	<input type="radio"/>

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

M system	<input type="radio"/>
L system	<input type="radio"/>

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 second 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

M system	<input type="radio"/>
L system	<input type="radio"/>

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

M system	<input type="radio"/>
L system	<input type="radio"/>

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.

- (1) Other operations
 - (a) When the retract signal is turned ON, the control unit is reset, the operation is interrupted, and the machining program is indexed.
 - (b) 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.
 - (c) 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.
 - (d) 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.
 - (e) 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.
 - (f) 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.

Program Support Functions

12.1 Machining Method Support Functions

12.1.1 Program

12.1.1.1 Subprogram Control

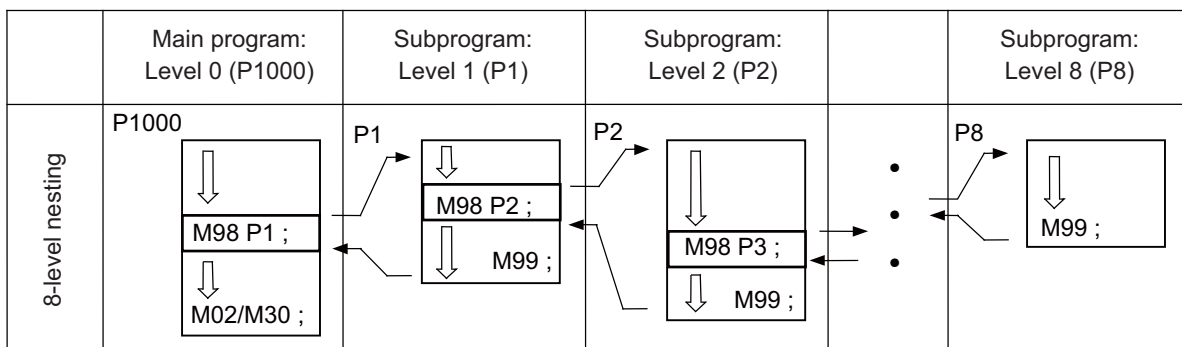
M system	○ 8 layers
L system	○ 8 layers

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.

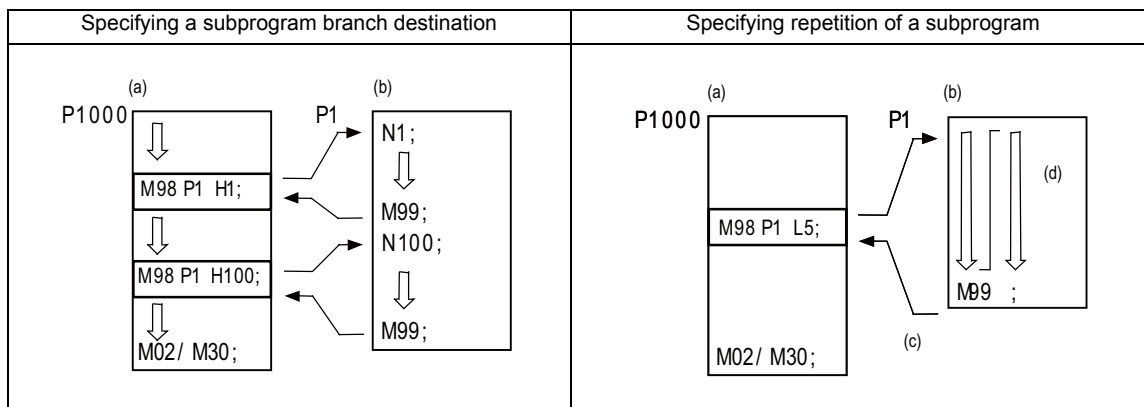
(1) Calling the subprogram with M98 command

M98 Pp1 Hh1 L1 ,Dd1 ;	
or, M98 <File name> Hh1 L1 ,Dd1 ;	
M98	: Subprogram call command
Pp1	: Program No. in subprogram to be called. (Own program if omitted.) Note that P can be omitted only during memory operation and MDI operation. (Max. 8-digit value)
<File name>	: A four-digit or eight-digit program No. starting with O No. can be called by the parameter. : A file name can be designated instead of the program No. In this case, enclose the file name with brackets <>. (The file name can have up to 32 characters, including the extension.) (Example) M98 <PARTS-12.RAF>;
Hh1	: Sequence No. in subprogram to be called (Head block if omitted) (Max. 6-digit value)
L1	: Number of subprogram repetitions (If omitted, this is handled as L1. When L0 is set, the subprogram is not called.) (1 to 9999 times set with 4-digit value.)
,Dd1	: Device No. where subprogram is stored. (0 to 4) If ,D is omitted, the subprogram in the memory will be used. The device No. is set with the machining parameters.

Subprograms can be nested up to eight levels deep.



A subprogram branch destination or repetition of a subprogram can be specified.



(2) Calling the subprogram with M198 command

A program registered in the front IC card (CF card) can be called out as a subprogram.

To call a program in the CF card as the subprogram, command as shown below in the main program.

M198 Pp1 Ll1 ; or, M198 <File name> Ll1 ;	
M198	: Call command
Pp1	: Program No. in CF card to be called as subprogram (Max. 8 digits)
<File name>	: A four-digit or eight-digit program No. starting with 0 No. can be called by the parameter. : A file name can be designated instead of the program No. In this case, enclose the file name with brackets <>. (The file name can have up to 32 characters, including the extension.)
Ll1	: Number of subprogram repetitions (Max. 4 digits) This can be omitted. (If omitted, this is handled as L1.) When L0 is set, the subprogram is not called.

(Note 1) Sequence No. call (M198 H****) is not possible.

(Note 2) Calling of the subprogram with the M198 command is limited to once in the subprogram nest.
The subprogram can be called only from the memory or MDI program.

(Note 3) The data from the head of the program to the first LF (line feed code. 0x0A as hexadecimal) is invalid, and cannot be run or displayed. Note that if the head starts from an O number (program No.), the program is valid from the head.

12.1.1.3 Scaling

M system	O
L system	—

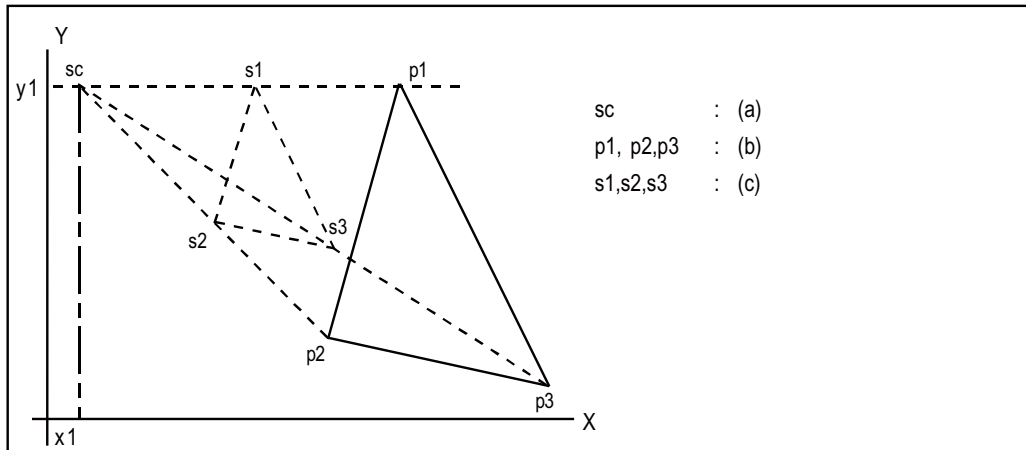
The shape commanded by the program can be extended or reduced to the desired size by applying a scale factor to the movement axis command position.

Gcode	Function
G50	Scaling cancel
G51	Scaling ON

The program format is given below.

```
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.



- (a) Scaling center
- (b) Programmed shape
- (c) Shape after scaling

- (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.)
- (Note 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.

12.1.2 Macro Program

12.1.2.1 User Macro

M system	O 4 layers
L system	O 4 layers

(1) 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 Pp1 LI1 --Argument--;or,G65 <File name> LI1 --Argument--;	
G65	: Call command
Pp1	: Program No.
<File name>	A four-digit or eight-digit program No. starting with O No. can be called by the parameter.
	: File name
LI1	When designating the file name, enclose the file name with brackets < >.
Argument	: Number of repetitions
	: Variable data assignment

The macro program is called immediately by this command

G66 Pp1 LI1 --Argument--;or,G66 <File name> LI1 --Argument--;	
G66	: Call command
Pp1	: Program No.
<File name>	A four-digit or eight-digit program No. starting with O No. can be called by the parameter.
	: File name
LI1	When designating the file name, enclose the file name with brackets < >.
Argument	: Number of repetitions
	: Variable data assignment

The macro program is executed from the block with the axis command following this command.

G66.1 Pp1 LI1 --Argument--;or, G66.1 <File name> LI1 --Argument--;	
G66.1	: Call command
Pp1	: Program No.
<File name>	A four-digit or eight-digit program No. starting with O No. can be called by the parameter.
	: File name
LI1	When designating the file name, enclose the file name with brackets < >.
Argument	: Number of repetitions
	: Variable data assignment

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.

(2) Macro commands 2

Specific G commands and the miscellaneous commands (M, S, T, B) can be used for macro call.

(a) Macro call using G codes

Simply by assigning a G code, it is possible to call user macro programs with the prescribed program number.

Format

G** Argument;	
G**	: G code for performing macro call
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.

Up to 10 codes from G100 to G999 can be used for this command. (The G codes that are used in the system also can be made available for this use by parameter setting.)

(Note 1) 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.

(b) 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.)

Format

M** (or S**, T**, B**);	
M** (S**, T**, B**)	: M (or S, T, B) code for performing macro call

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. Up to 10 M codes from M00 to M9999, M198 can be entered. Select codes to be entered other than the codes basically required by the machine and M codes of M0, M1, M2, M30 and M96, M99 through M198.

(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.)

12.1.2.3 Macro Interruption

M system	○
L system	○

By inputting a user macro interrupt signal from the PLC, the program being currently executed is interrupted and other programs can be called instead.

Retract or return operations when tools have been damaged, for instance, and other kinds of restoration operations to be conducted when trouble has occurred are programmed in the interrupt programs.

There are two types of interrupts, type 1 and type 2, as described below, and they are selected using a parameter.

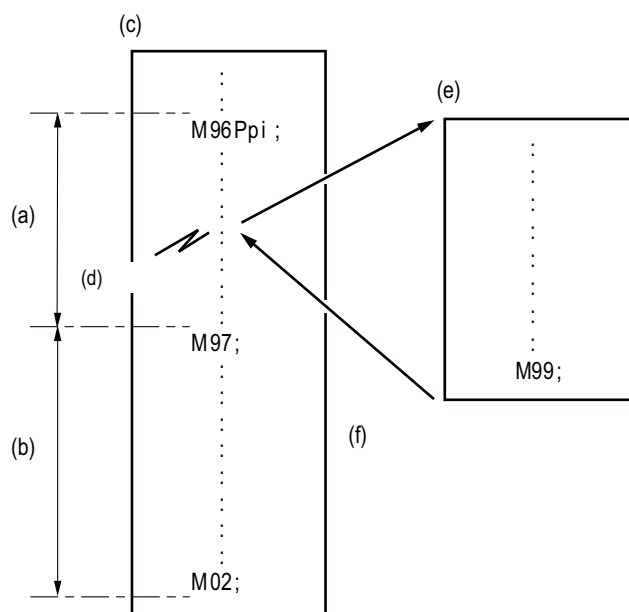
[Interrupt type 1] The block being executed is immediately interrupted, and the interrupt program is run immediately.

[Interrupt type 2] After the block being executed is complete, the interrupt program is executed.

The command format is given below.

M96 Pp1 Hh1 ; or, M96 <File name> Hh1 ; User macro interrupt valid	
M96	: Interrupt valid command
Pp1	: Interrupt program number
<File name>	: A four-digit or eight-digit program No. starting with O No. can be called by the parameter.
	: File name
Hh1	: When designating a file name, enclose the file name in brackets <>.
	: Interrupt sequence No.

M97 ; User macro interrupt invalid	
M97	: User macro interrupt end command



- (a) The user macro interrupt signal is accepted during this period.
- (b) The user macro interrupt signal is not accepted during this period.
- (c) Machining program Opm:
- (d) Interrupt signal
- (e) Interrupt program Opi
- (f) The modal information is restored to the status applying before interrupt.

12.1.2.4 Variable Command

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 variables	Common variables 1	Common variables 2	Can be used commonly for main, sub and each macro program.
	200 sets	500 to 599 100 to 199	
Local variables	1 to 33		Can be used as local in macro program.
System variables	1000 to		Application is fixed in system.
Fixed cycle variables	1 to 32		Local variables in fixed cycle program.

(Note 1) All common variables are held even when the power is turned OFF.

(Note 2) The common variables can be emptied by resetting or turning the power OFF when the parameters are set accordingly.

(Note 3) Variable names can be set for #500 to #519

Variable expressions

Variable : #Numerical value #100

(Numerical value: 1,2,3,)

: # [Expression] #100

Expression : Numerical value

: Variable

: Expression Operator Expression #100 + #101

: - (minus) Expression -#120

: [Expression] [#110]

: Function [Expression] SIN [#110]

Variable definition

Variable = expression

(Note1) Variables cannot be used with addresses "O" and "N".

12.1.2.4.2 200 Sets

M system	○
L system	○

12.1.3 Fixed Cycle

List of fixed cycles

Type of fixed cycle	M system	L system				Remarks
	Gcode list 1	Gcode list 2	Gcode list 3	Gcode list 6	Gcode list 7	
Fixed cycle for drilling	G70	G80	G80	G80	G80	Refer to 12.1.3.1. Refer to 12.1.3.2 (Type II). Refer to 4.5.3.
	:	:	:	:	:	
	G89	G89	G89	G89	G89	
		G79	G83.2	G79	G83.2	
	G98	G98	G98	G98	G98	
	G99	G99	G99	G99	G99	
Special fixed cycles	G34					Refer to 12.1.3.3.
	G35	-	-	-	-	
	G36					
Fixed cycles for turning machining		G90	G77	G90	G77	Refer to 12.1.3.4.
	-	G92	G78	G92	G78	
		G94	G79	G94	G79	
Compound-type fixed cycles for turning machining		G70	G70	G70	G70	Refer to 12.1.3.5. Refer to 12.1.3.6 (Type II).
		:	:	:	:	
	-	G76	G76	G76	G76	

12.1.3.1 Fixed Cycle for Drilling

M system	<input type="radio"/>
L system	<input type="radio"/>

(1) M system: G70 to G89, G88, G99

These functions enable drilling, tapping and other hole machining cycles to be assigned in a simple 1-block program.

Gcode	Function
G70	
G71	
G72	
G73	Step cycle
G74	Reverse tapping cycle
G75	Perfect circle cutting cycle
G76	Fine boring
G77	
G78	
G79	
G80	Fixed cycle cancel
G81	Drilling, spot drilling cycle
G82	Drilling, counterboring cycle
G83	Deep hole drilling cycle
G84	Tapping cycle
G85	Boring cycle
G86	Boring cycle
G87	Backboring cycle
G88	Boring cycle
G89	Boring cycle

There are two levels of hole machining axis return which apply upon completion of the fixed cycle machining operation.

Gcode	Function
G98	Initial point level return
G99	R point level return

The basic program format for the fixed cycle commands is shown below.

G81 Xx1 Yy1 Zz1 Rr1 Qq1 Pp1 Ll1 Ff1 ;	
G81	: Hole drilling mode
Xx1, Yy1	: Hole position data; X-axis, Y-axis hole drilling position command (rapid traverse) (incremental/absolute)
Zz1	: Hole machining data; Hole bottom position designation (incremental/absolute)
Rr1	: Hole machining data; Hole R point designation (incremental/absolute)
Qq1	: Hole machining data; Depth of cut per pass in G73, G83 cycle (incremental) Shift amount in G76, G87 cycle
Pp1	: Hole machining data; Dwell time at hole bottom
Ll1	: Hole machining data; Number of fixed cycle repetitions (Not executed when "0" is set.)
Ff1	: Cutting feed rate, Z-axis feed amount (tapping pitch) per spindle rotation

For details on the synchronous tapping cycle, refer to the section "4.5.3 Synchronous Tapping".

<p>G73 Step cycle</p> <p>Initial point R point Z point G98 mode G99 mode</p>	<p>G74 Reverse tapping cycle</p> <p>Initial point R point Z point G98 mode M04 M03</p>	<p>G75 Perfect circle cutting cycle</p> <p>Initial point R point Z point G98 mode G99 mode</p>	<p>G76 Fine boring cycle</p> <p>Initial point R point Z point G98 mode M19 Shift G99 mode</p>
<p>G81 Drilling, spot drilling cycle</p> <p>Initial point R point Z point G98 mode G99 mode</p>	<p>G82 Drilling, counterboring cycle</p> <p>Initial point R point Z point G98 mode G99 mode Dwell</p>	<p>G83 Deep hole drilling cycle</p> <p>Initial point R point Z point G98 mode G99 mode</p>	<p>G84 Tapping cycle</p> <p>Initial point R point Z point G98 mode M03 M04</p>
<p>G85 Boring cycle</p> <p>Initial point R point Z point G98 mode</p>	<p>G86 Boring cycle</p> <p>Initial point R point Z point M03 G98 mode M05</p>	<p>G87 Back boring cycle</p> <p>Initial point R point Z point M19 M03</p>	<p>G88 Boring cycle</p> <p>Initial point R point Z point M03 M05 G98 mode Dwell</p>
<p>G89 Boring cycle</p> <p>Initial point R point Z point G98 mode Dwell</p>			

II Functional Specifications

(2) L system: G83 to G89, G80

These functions enable drilling, tapping and other hole machining cycles to be assigned in a simple 1-block program.

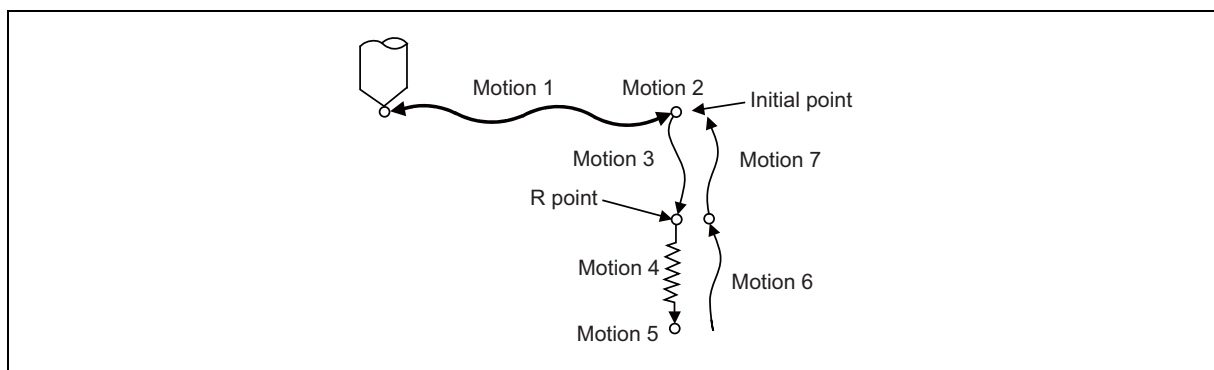
Gcode	Drilling axis	Drilling work start	Motion at hole bottom	Retract motion	Use
G80					Cancel
G83	Z	Cutting feed Intermittent feed	In-position check Dwell	Rapid traverse feed	Deep-hole drilling cycle1
G84	Z	Cutting feed	In-position check Dwell Spindle CCW	Cutting feed	Tapping cycle (Reverse tapping cycle)
G85	Z	Cutting feed	In-position check Dwell	Cutting feed	Boring cycle
G87	X	Cutting feed Intermittent feed	In-position check Dwell	Rapid traverse feed	Deep-hole drilling cycle1
G88	X	Cutting feed	In-position check Dwell Spindle CCW	Cutting feed	Tapping cycle (Reverse tapping cycle)
G89	X	Cutting feed	In-position check Dwell	Cutting feed	Boring cycle
G83.2	Z/X	Cutting feed Intermittent feed	In-position check Dwell	Rapid traverse feed	Deep-hole drilling cycle2

The fixed cycle mode is canceled when G80 or some G command of the G01 group is issued. Data is also cleared to zero simultaneously.

Command format

G83/G84/G85 Xx1 Cc1 Zz1 Rr1 Qq1 Pp1 Ff1 Kk1(Mm1) Ss1 ,Ss1 Dd1 ,Rr1 ;	
G87/G88/G89 Xx1 Cc1 Zz1 Rr1 Qq1 Pp1 Ff1 Kk1(Mm1) Ss1 ,Ss1 Dd1 ,Rr1 ;	
G83/G84/G85 G87/G88/G89	: Fixed cycle mode of drilling (G83, G87), tapping (G84, G88), or boring (G85, G89) These commands are modal. Once one of them is given, it is effective until another command of the same modal group or cancel command for drilling fixed cycle is given.
Xx1, Cc1	: Data for positioning X (Z) and C axes The data is unmodal. To execute the same hole machining mode consecutively, specify the data for each block.
Zz1, Rr1, Qq1, Pp1, Ff	: Actual machining data in machining Only Q is unmodal. Specify Q in G83 or G87 for each block whenever the data is required.
Kk1	: To repeat in a single cycle for hole machining at equal intervals, specify the number of repetitions in the range of 0 to 9999 (no decimal point can be used). It is unmodal and is effective only in the block in which the number of repetitions is specified. If the number of repetitions is omitted, K1 is assumed to be specified. If K0 is specified, hole machining data is stored, but hole machining is not performed.
Mm1	: If axis C clamp M command (parameter setting) is given, the M code is output at the initial point, and after return motion, C axis unclamp M code (clamp M code + 1) is output and the dwell time set in a given parameter is executed.
Ss1	: Designates spindle rotation speed
,Ss1	: Designates spindle rotation speed at retract
Dd1	: Designates tapping spindle No. for G84 (G88)
,Rr1	: Changes between synchronous/asynchronous in G84 (G88)

The drilling cycle motions generally are classified into the following seven.



Motion 1: Rapid positioning up to the initial point of X (Z) and C axes.

If the "positioning axis in-position width" is designated, the in-position check is conducted upon completion of the block.

Motion 2: Output if the C axis clamp M code is given.

Motion 3: Rapid positioning up to the R point.

Motion 4: Hole machining at cutting feed.

If the "drilling axis in-position width" is designated, the in-position check is conducted upon completion of the block. However, in the case of deep-hole drilling cycles 1 and 2, the in-position check is not conducted with the drilling of any holes except the last one. The in-position check is conducted at the commanded hole bottom position (last hole drilling).

Motion 5: Motion at the hole bottom position. It varies depending on the fixed cycle mode.

Spindle CCW (M04), spindle CW (M03), dwell, etc., are included.

Motion 6: Return to the R point.

Motion 7: Return to the initial point at rapid traverse feed.

(Motions 6 and 5 may be conducted as a single motion depending on the fixed cycle mode.)

(Note) With a synchronous tap command, the in-position check is conducted in accordance with the parameters.

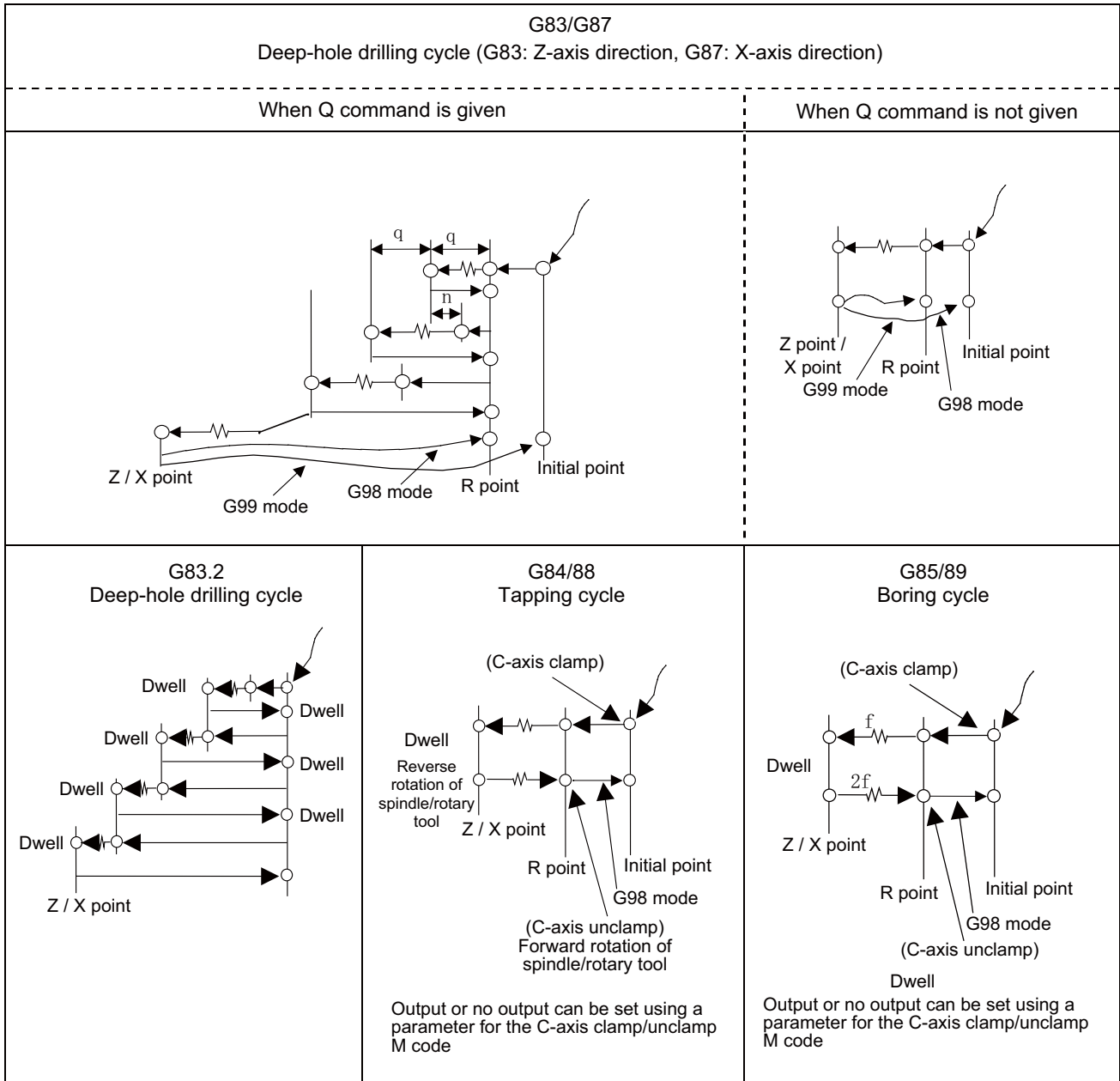
Whether the fixed cycle is complete with motion 6 or 7 can be specified by using either of the following G commands:

G98 : Initial level return

G99 : R point level return

These commands are modal. For example, once G98 is given, the G98 mode is entered until G99 is given. The G98 mode is entered in the initial state when the controller is ready.

Deep-hole drilling cycle (G83, G87)



There are two levels of hole machining axis return which apply upon completion of the fixed cycle machining operation.

Gcode	Function
G98	Initial point level return
G99	R point level return

12.1.3.2 Fixed Cycle for Drilling (Type II)

M system	—
L system	O

In the longitudinal hole drilling fixed cycle, the X axis is designated as the hole drilling axis. However, in the longitudinal hole drilling fixed cycle (type II), the Y axis can be designated as the hole drilling axis with the longitudinal hole drilling axis selection function.

The relationship between the longitudinal hole drilling axis selection signal's ON/OFF state and the hole drilling axis of the fixed cycle for drilling is shown below.

Gcode	Details	Y axis cross tap function selection signal state	Hole drilling axis
G80	Cancel	-	-
G83	Deep hole drilling cycle 1	ON	Z
		OFF	
G84 (G84.1)	Tapping cycle	ON	Z
		OFF	
G85	Boring cycle	ON	Z
		OFF	
G87	Deep hole drilling cycle 1	ON	Y
		OFF	X
G88 (G88.1)	Tapping cycle	ON	Y
		OFF	X
G89	Boring cycle	ON	Y
		OFF	X
G83.2	Deep hole drilling cycle 2	ON	Z/X
		OFF	

12.1.3.3 Special Fixed Cycle

M system	O
L system	—

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:

Gcode	Function
G34	Bolt hole circle
G35	Line at angle
G36	Arc
G37.1	Grid

(1) 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; this is affected by the G90/G91 commands.
 Ir : Radius "r" of circle; it is based on the least command increment and is provided using a positive number.
 Jθ : Angle θ at 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.

(Example)
 With 0.001mm least command increment
 N001 G91 ;
 N002 G81 Z-10.000 R5.000 L0 F200 ;
 N003 G90 G34 X200.000 Y100.000 I100.000 J20.000 K6 ;
 N004 G80 ;.....(G81 cancel)
 N005 G90 G0 X500.000 Y100.000 ;

- (a) G0 command in N005
- (b) Position prior to execution of G34 command
- (c) n = 6 holes

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.

(2) 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.

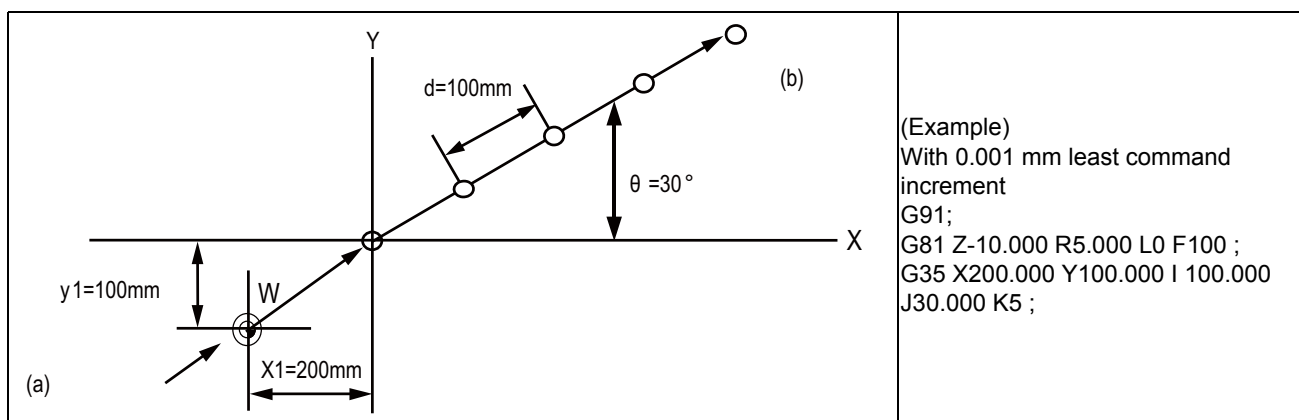
G35 Xx Yy Id J θ Kn ;

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.

J θ : Angle θ ; the counterclockwise direction is taken to be positive.

Kn : Number "n" of holes to be drilled including the starting point; any number of holes from 1 through 9999 can be assigned.

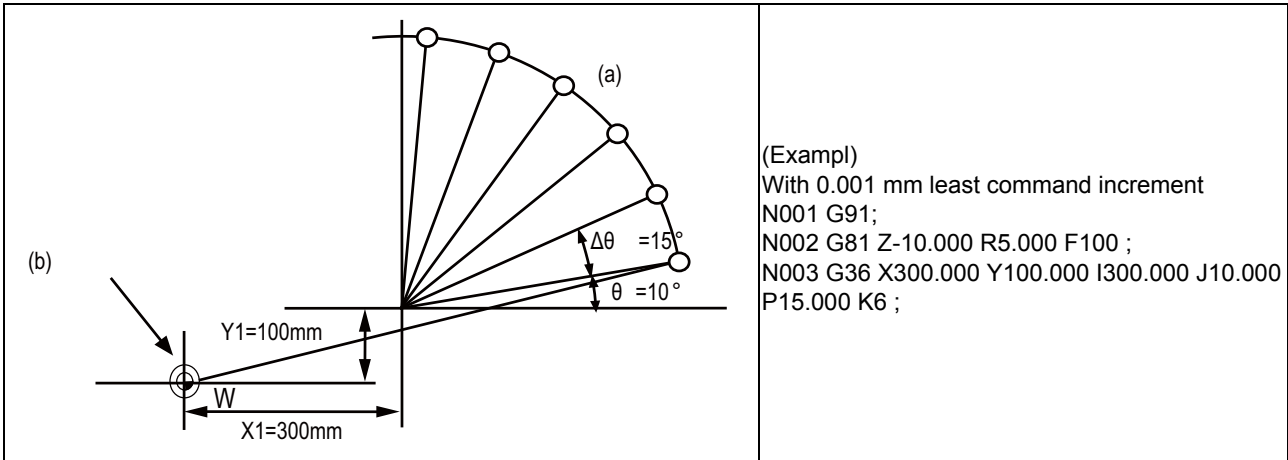


(3) Arc (G36)

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 $\Delta\theta$. 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 P $\Delta\theta$ 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.
J θ	: Angle θ at the point to be drilled initially; the counterclockwise direction is taken to be positive.
P $\Delta\theta$: Angle interval $\Delta\theta$; 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.



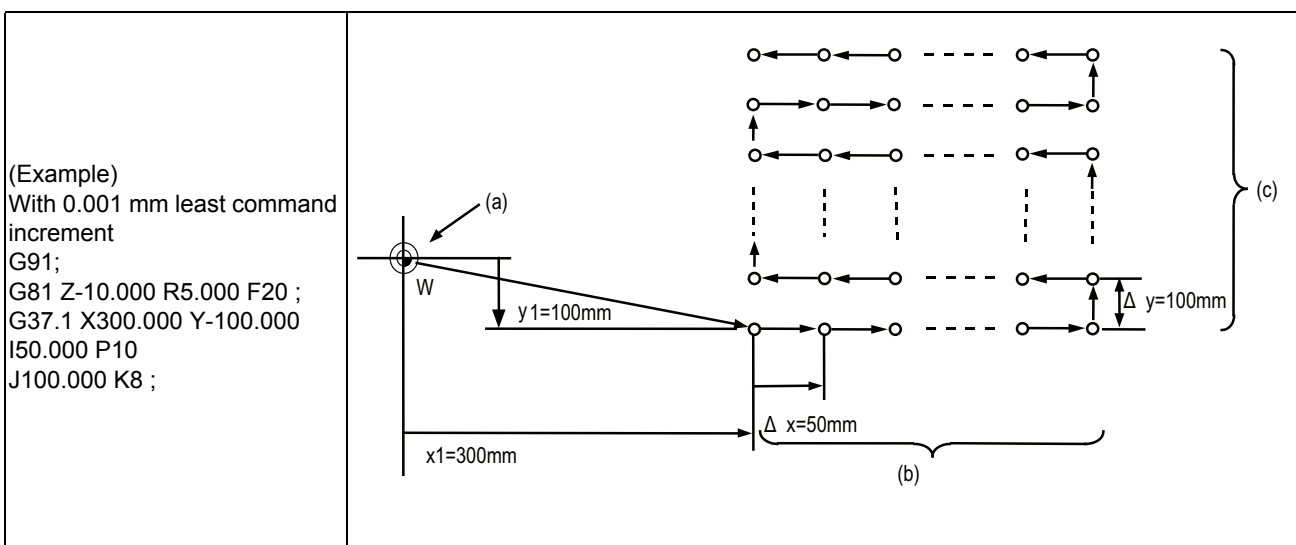
(Exempl)
 With 0.001 mm least command increment
 N001 G91;
 N002 G81 Z-10.000 R5.000 F100 ;
 N003 G36 X300.000 Y100.000 I300.000 J10.000
 P15.000 K6 ;

(a) n=6 holes
 (b) Position prior to execution of G36 command

(4) 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 I Δx Pnx J Δy 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; any number of holes from 1 through 9999 can be assigned.
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; any number of holes from 1 through 9999 can be assigned.



- (a) Position prior to execution of G37.1 command
- (b) nx=10 holes
- (c) ny=8 holes

12.1.3.4 Fixed Cycle for Turning Machining

M system	—
L system	O

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. The fixed cycles are as follows.

G code	Function
G77	Longitudinal cutting cycle
G78	Thread cutting cycle
G79	Face cutting cycle

Format

G Δ Δ X/U __ Z/W __ I __ K __ R __ F __ (G18 plane)
--

Each fixed cycle command for turning machining is a modal G code and is effective until another command of the same modal group or a cancel command is given.

The fixed cycle can be canceled by using any of the following G codes:

G00,G01,G02,G03

G09

G10,G11

G27,G28,G29,G30

G31

G33,G34

G37

G92

G52,G53

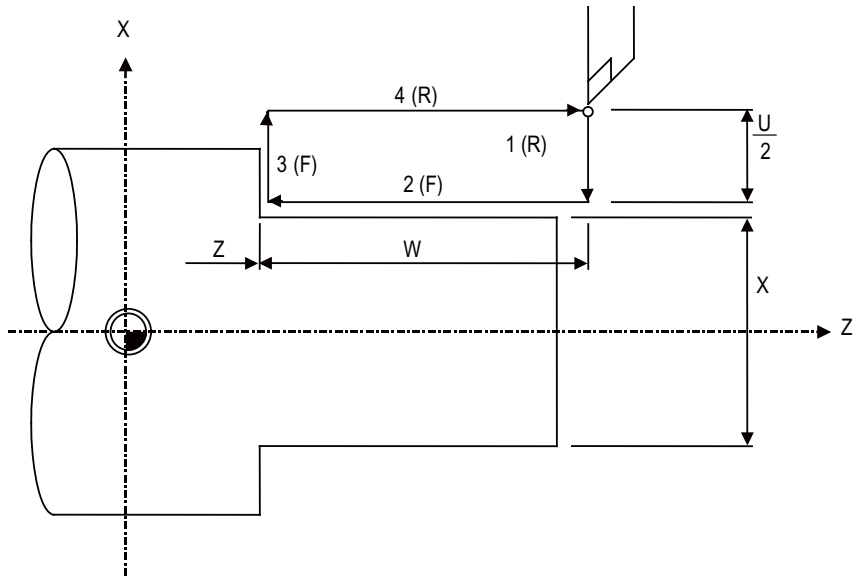
G65

(1) Longitudinal cutting cycle (G77)

(a) Straight cutting

Straight cutting in the longitudinal direction can be performed consecutively by the following block:

```
G77 X/U_Z/W_F_;
```

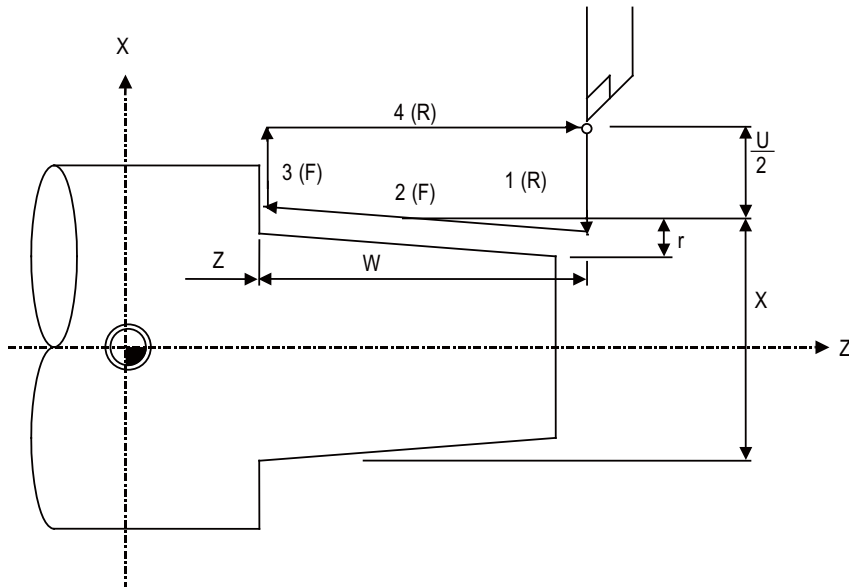


(F) Cutting feed
(R) Rapid traverse feed

(b) Taper cutting

Taper cutting in the longitudinal direction can be performed consecutively by the following block:

```
G77 X/U_Z/W_R_F_;
```



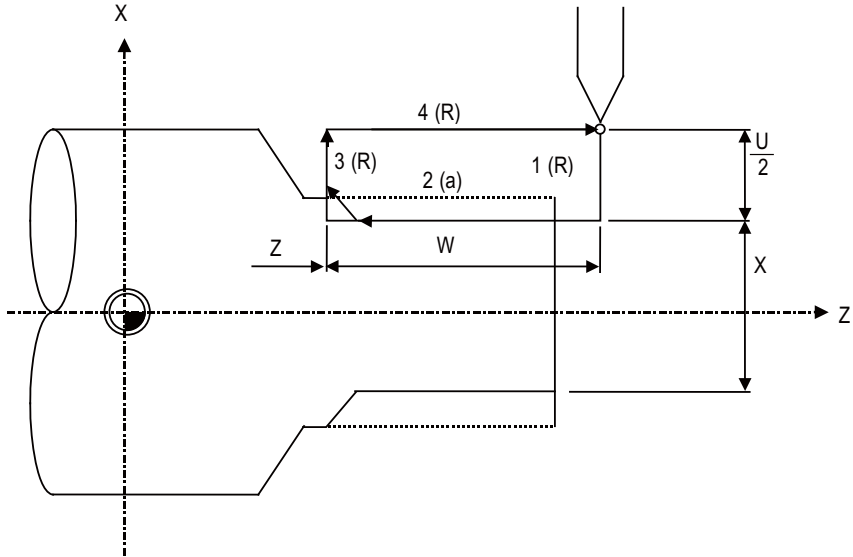
(F) Cutting feed
(R) Rapid traverse feed
r: Taper part depth (radius designation, incremental position, sign is required)

(2) Thread cutting cycle (G78)

(a) Straight thread cutting

Straight thread cutting can be performed by the following block:

```
G78 X/U_ Z/W_ F/E_ ;
```

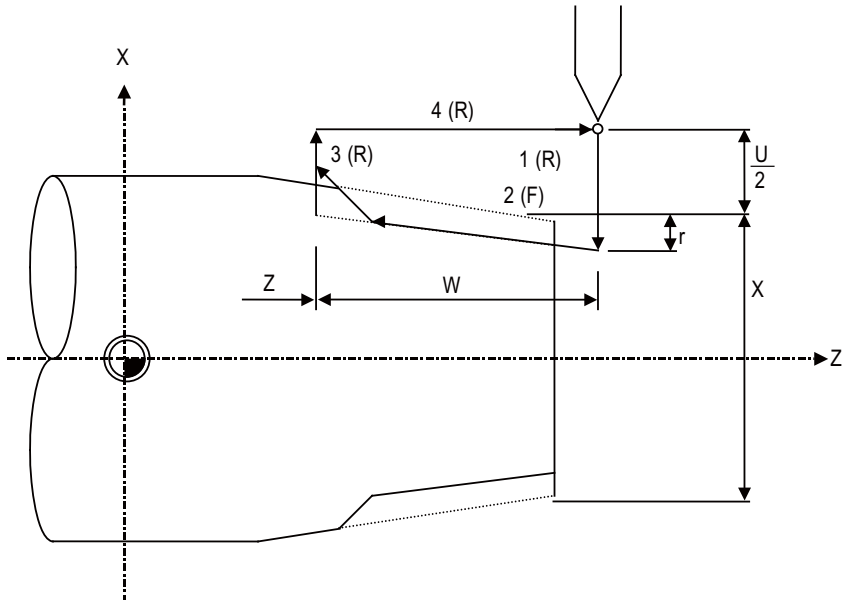


- (a) F or E code designation
- (R) Rapid traverse feed

(b) Taper thread cutting

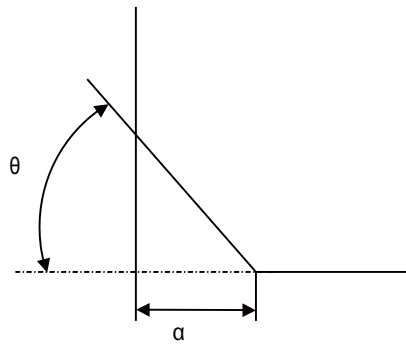
Taper thread cutting can be performed by the following block:

```
G78 X/U_ Z/W_ R_ F/E_ ;
```



- (R) Rapid traverse feed
- (F) F or E code designation
- r : Taper part depth (radius designation, incremental position, sign is required)

Chamfering



α : Thread cutting-up amount

Assuming that thread lead is L, the thread cutting-up amount can be set in a given parameter in 0.1L steps in the range of 0 to 12.7L.

θ : Thread cutting-up angle

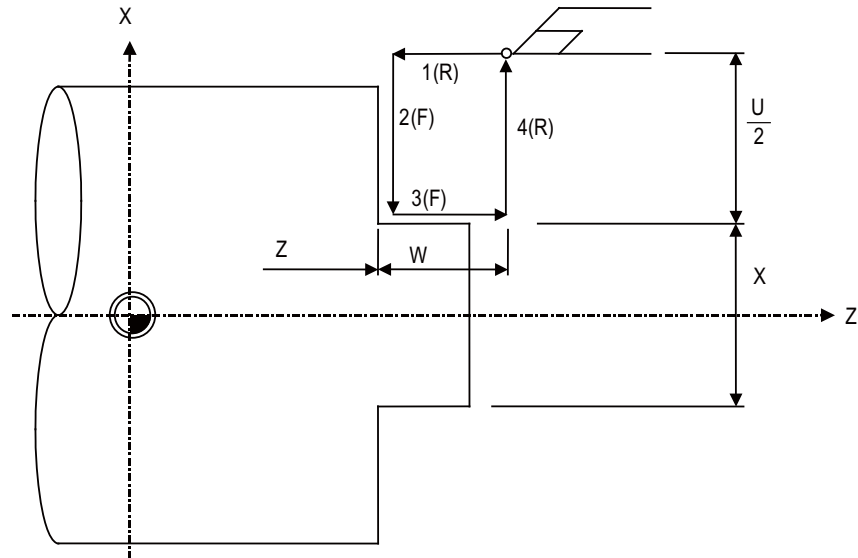
The thread cutting-up angle can be set in a given parameter in 1° steps in the range of 0 to 89°

(3) Face cutting cycle (G79)

(a) Straight cutting

Straight cutting in the end face direction can be performed consecutively by the following block:

```
G79 X/U_ Z/W_ F_ ;
```

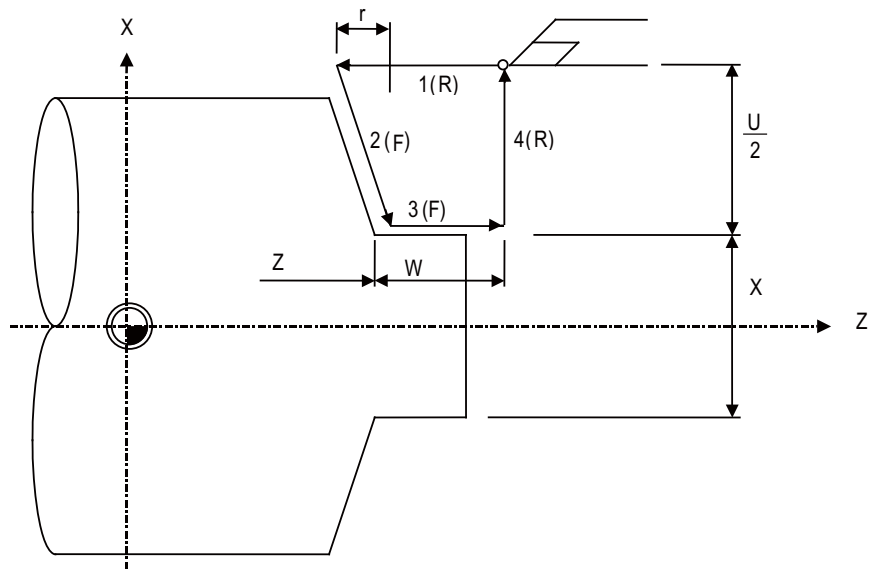


(R) Rapid traverse feed
(F) Cutting feed

(b) Taper cutting

Taper cutting in the end face direction can be performed consecutively by the following block:

```
G79 X/U_ Z/W_ R_ F_ ;
```



(R) Rapid traverse feed
(F) Cutting feed
r: Taper part depth (radius designation, incremental position, sign is required)

12.1.3.5 Compound Type Fixed Cycle for Turning Machining

M system	—
L system	O

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:

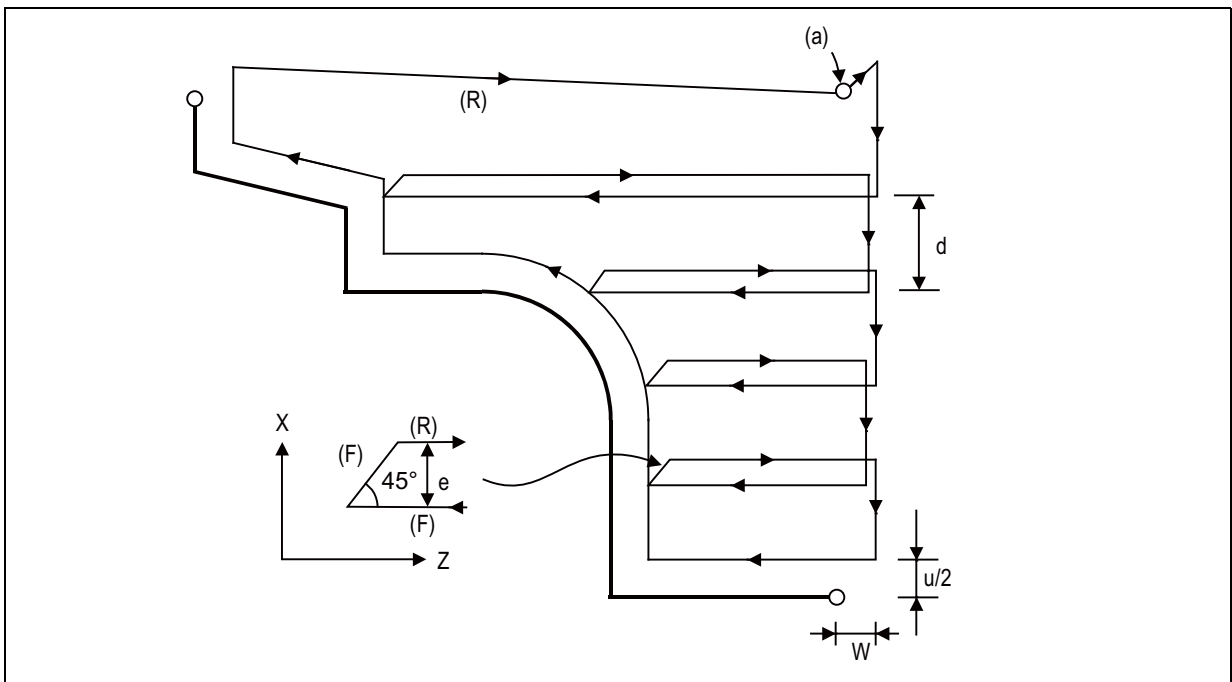
G code	Function
G71	Longitudinal rough cutting cycle
G72	Face rough cutting cycle
G73	Molding material in rough cutting cycle
G70	Finish cycle
G74	Face cutting-off cycle
G75	Longitudinal cutting-off cycle
G76	Multiple repetitive thread cutting cycle

(1) Longitudinal rough cutting cycle (G71)

The finish shape program is called, and straight rough cutting is performed while intermediate path is being calculated automatically.

The machining program is commanded as follows.

G71 Ud Re ;	
G71 Aa Pp Qq Uu Ww Ff Ss Tt ;	
Ud	: Cutting amount (modal) Reversible parameter Increment : μm or 1/10000inch ... Radius value command
Re	: Retract amount (modal) Reversible parameter Increment : μm or 1/10000inch ... Radius value command
Aa	: Finish shape program No. (If omitted, the program being executed is designated.) A four-digit or eight-digit program No. starting with O no. can specify the finish shape program by the parameter. If the setting is to call the program with O No., it becomes the A command value program starting with O No. If the A command is omitted, the program being executed are applied. If A is omitted, the program following the end of this cycle will be executed at the block after Qq (finish shape end sequence No.). A file name can be designated instead of address A by enclosing the file name in brackets <>. (The file name can have up to 32 characters, including the extension.)
Pp	: Finish shape start sequence number (Head of program if omitted.)
Qq	: Finish shape end sequence number (To end of program if omitted.) 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 1/10000inch Diameter/radius value command follows changeover parameter.
Ww	: Finishing allowance in Z axis direction (If omitted, finishing allowance in Z axis direction is handled as 0.) Increment : μm or 1/10000inch Radius value command
Ff	: Cutting speed (If omitted, cutting speed (modal) before G71 is applied.)
Ss, Tt	: Spindle command, tool command



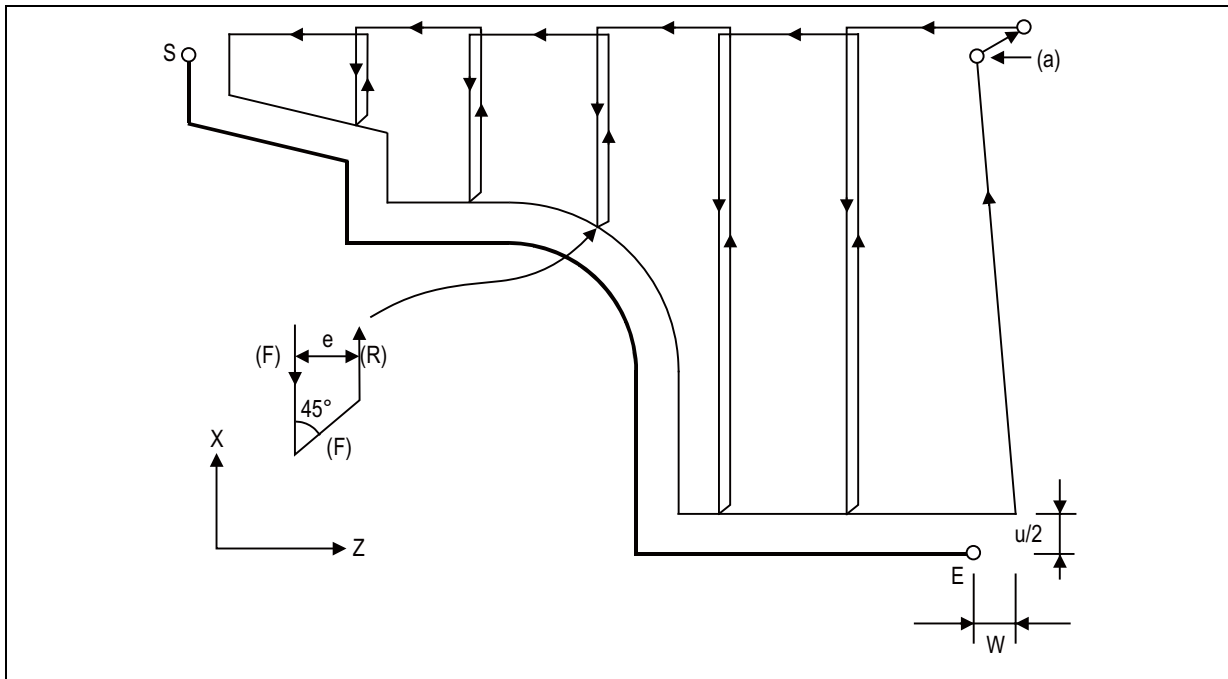
- (a) (Cycle commanded point)
- d Cutting amount
- u/2 Finishing allowance
- W Finishing allowance
- (R) Rapid traverse feed
- (F) Cutting feed

(2) 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.

The machining program is commanded as follows

G72 Wd Re ;	
G72 Aa Pp Qq Uu Ww Ff Ss Tt ;	
	: Cut depth d. (When P,Q command is not given). (Modal)
	: Retract amount e. (Modal)
Wd	: Finish shape program No. (If it is omitted, the program being executed is designated.)
Re	: A four-digit or eight-digit program No. starting with O no. can specify the finish shape program by the parameter.
Aa	: If the setting is to call the program with O No., it becomes the A command value program starting with O No. : If the A command is omitted, the P, Q commands in the program being executed are applied. : If the A command is omitted, the program being executed are applied. : If A is omitted, the program following the end of this cycle will be executed at the block after Qq (finish shape end sequence No.).
Pp	: A file name can be designated instead of address A by enclosing the file name in brackets <>.
Qq	: (The file name can have up to 32 characters, including the extension.)
Uu	: Finish shape start sequence number (Head of program if omitted.) : Finish shape end sequence number (To end of program if omitted.) : If M99 precedes the Q command, up to M99.
Ww	: Finishing allowance in X axis direction (If omitted, finishing allowance in X axis direction is handled as 0.)
Ff	: Finishing allowance in Z axis direction
Ss, Tt	: (If omitted, finishing allowance in Z axis direction is handled as 0.) : Cutting feedrate (If omitted, cutting feedrate (modal) before G72 is applied.) : Spindle command, tool command



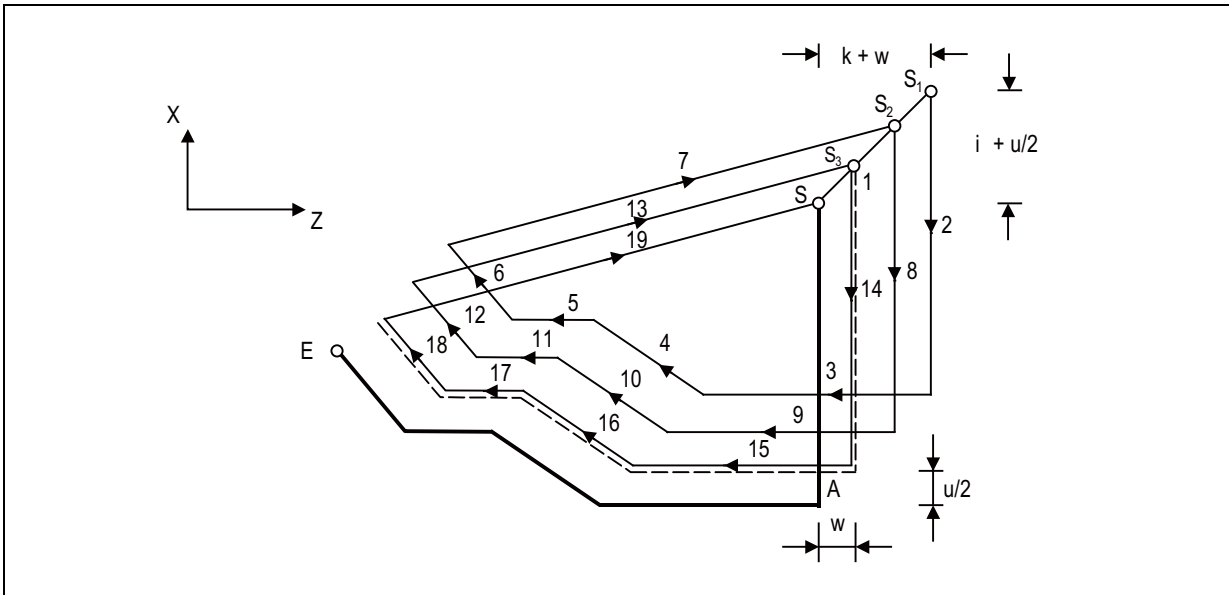
- (a) Cycle commanded point
- d Cut depth
- u/2 Finishing allowance
- W Finishing allowance
- (F) Cutting feed
- (R) Rapid traverse feed

(3) Molding material in 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.

The machining program is commanded as follows.

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.
Wk	:Cutting allowance in the Z axis direction k	- Modal data
Rd	:Split count d	- Sign is ignored.
		- Cutting allowance is given with a radius designation.
		(If it is omitted, the present program is assumed to be designated.)
Aa	:Finish shape program No.	A file name can be designated instead of address A by enclosing the file name in brackets <>. (The file name can have up to 32 characters, including the extension.)
		(If it is omitted, the program top is assumed to be designated.)
Pp	:Finish shape start sequence No.	(If it is omitted, the program end is assumed to be designated.)
Qq	:Finish shape end sequence No.	However, if M99 precedes the Q command, up to M99.
		- Finishing allowance for the finish shape commanded by the address P or Q.
Uu	:Finishing allowance in the X axis direction u	- Sign is ignored.
Ww	:Finishing allowance in the Z axis direction w	- Diameter or radius is designated according to the parameter.
		- The shift direction is determined by the shape
Ff	:Cutting feed rate (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 selection (T function)	



(4) Finish cycle (G70)

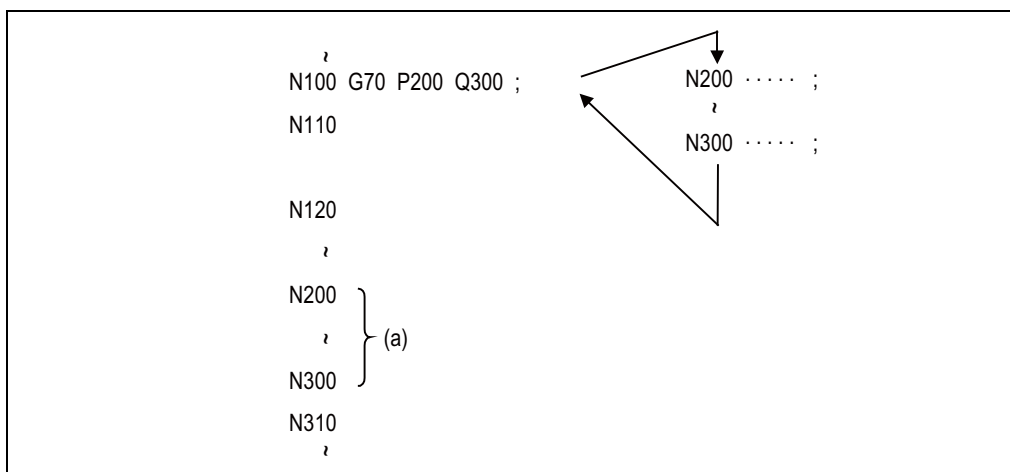
After rough cutting is performed by using G71 to G73, finish cutting can be performed by using the G70 command.

The machining program is commanded as follows.

G70 Aa Pp Qq ;	
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 brackets <>. (The file name can have up to 32 characters, including the extension.)
Pp	: Finish shape start sequence No. (If it is omitted, the program top is assumed to be designated.)
Qq	: Finish 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.

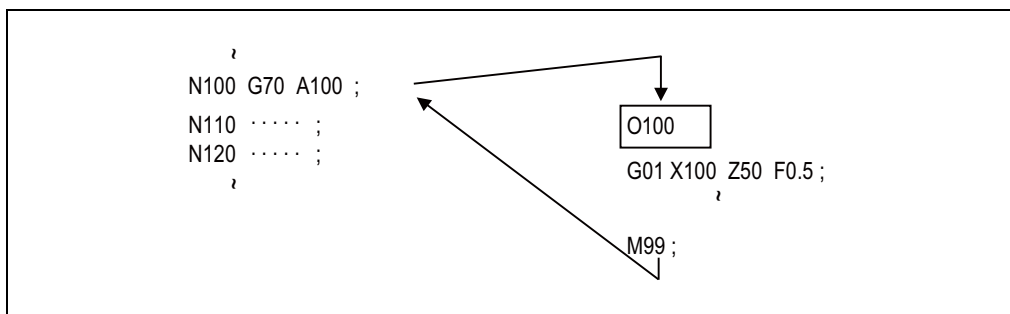
- (a) The F, S, and T commands in the rough cutting cycle command G71 to G73 blocks are ignored, and the F, S, and T commands in the finish shape program become effective.
- (b) The memory address of the finish shape program executed by G71 to G72 is not stored. Whenever G70 is executed, a program search is made.
- (c) When the G70 cycle terminates, the tool returns to the start point at the rapid traverse feed rate and the next block is read.

(Example1) Sequence No. designation



(a) Finish shape program

(Example2) Program No. designation



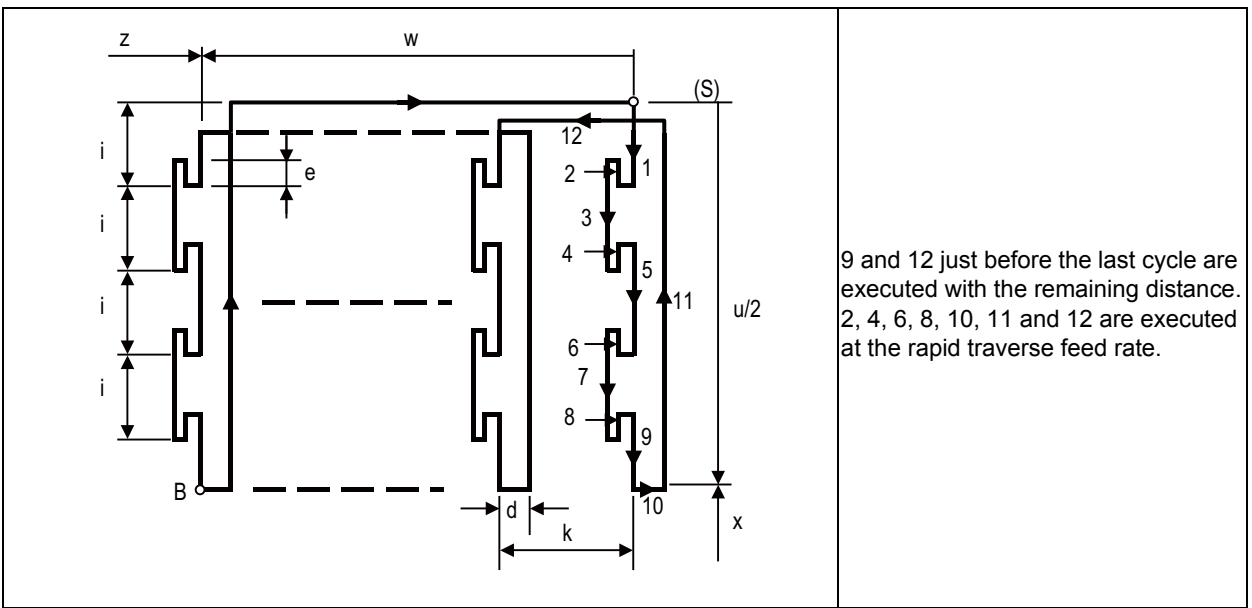
In either example 1 or 2, after the N100 cycle is executed, the N110 block is executed.

(5) Face cutting-off cycle (G74)

When the slotting end point coordinates, cut 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. The machining program is commanded as follows.

```

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      : Relief amount at cut bottom d (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      : Feed rate
    
```



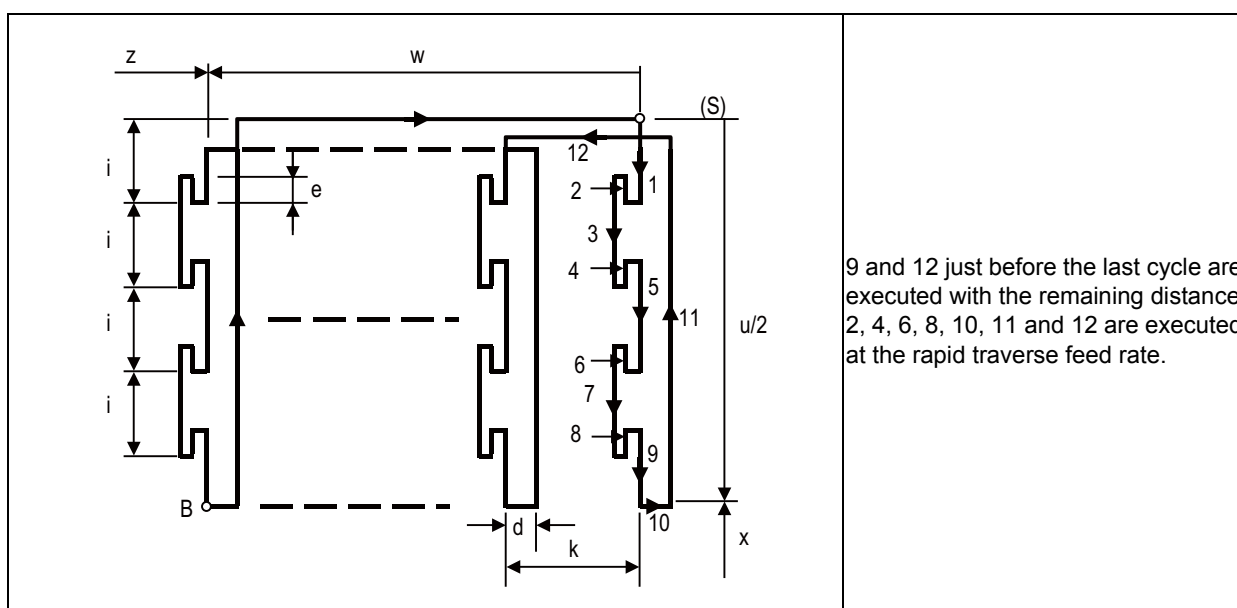
9 and 12 just before the last cycle are executed with the remaining distance. 2, 4, 6, 8, 10, 11 and 12 are executed at the rapid traverse feed rate.

(S) Start point

(6) Longitudinal cutting-off cycle (G75)

When the slotting end point coordinates, cut 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. The machining program is commanded as follows.

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	: Relief amount at cut bottom d (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	: Feed rate



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.

(S) Start point

(7) Compound type thread cutting cycle (G76)

When the thread cutting start and end points are commanded, cut at any desired angle can be made by automatic cutting so that the cut section area (cutting torque) per time becomes constant in the G76 fixed cycle.

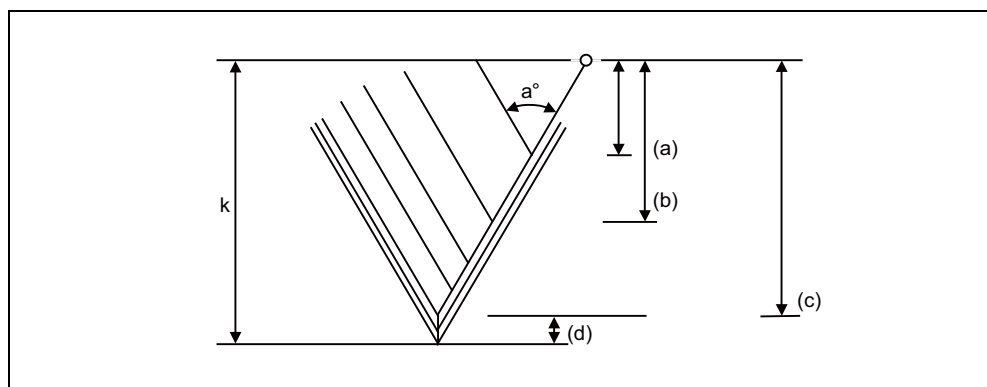
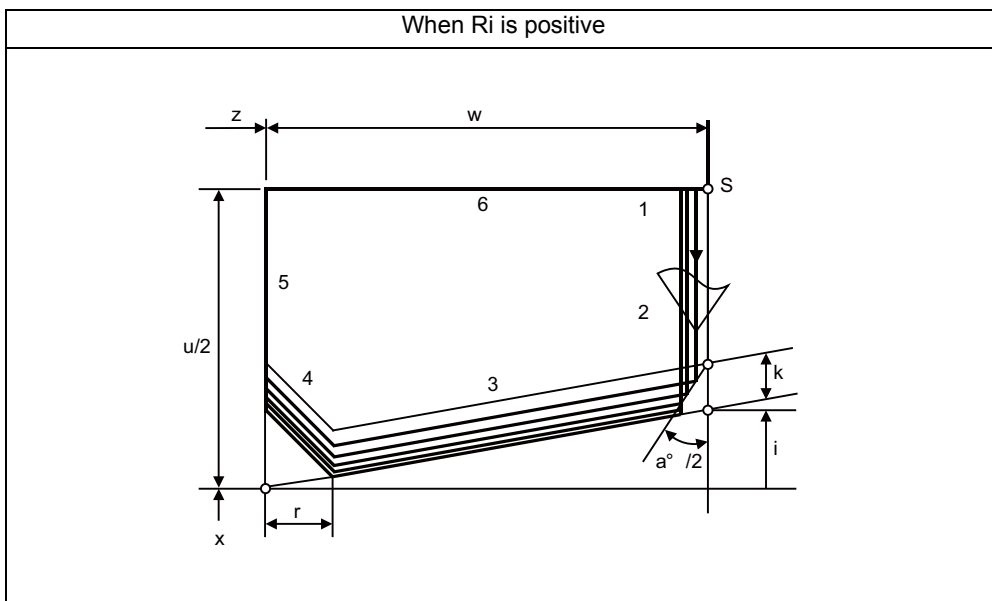
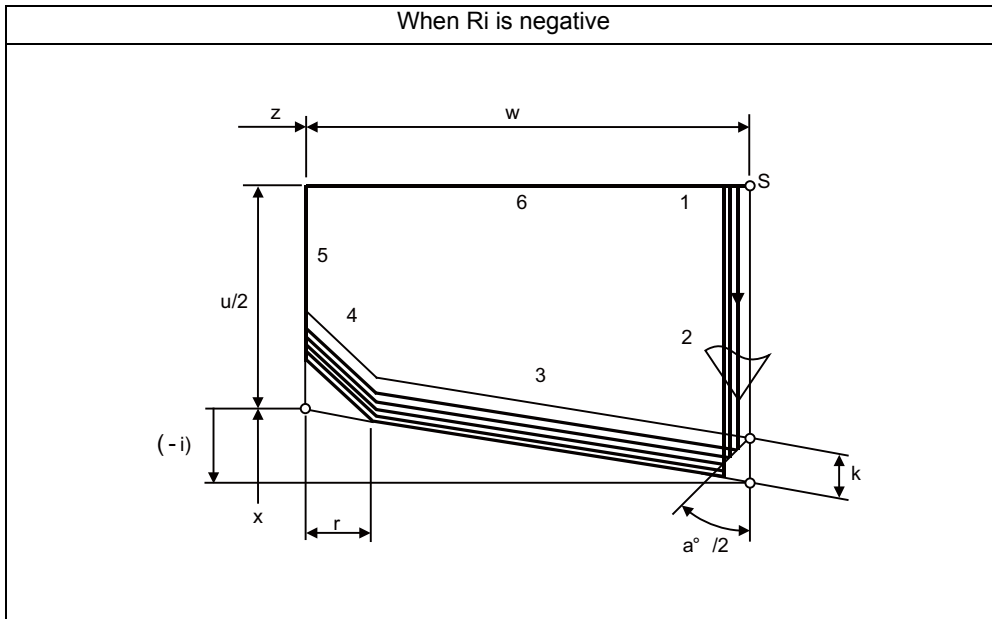
Various longitudinal threads can be cut by considering the thread cutting end point coordinate and taper height constituent command value.

Command Format

G76 Pmra Q Δ dmin Rd ;	
G76 Xx1/Uu1 Zz1/Ww1 Ri Pk Q Δ d FI ;	
m	: Cut count at finishing 01 to 99 (modal)
r	: Chamfering amount 00 to 99 (modal). Set in 0.1-lead increments.
a	: Nose angle (included angle of thread) 00 to 99 (modal) Set in 1-degree increments.
Δ dmin	: Least cut depth When the calculated cut depth becomes smaller than Δ dmin, the cut depth is clamped at Δ dmin.
d	: Finishing allowance (modal)
Xx1/Uu1	: X axis end point coordinate of thread part. --- Absolute/Incremental position
Zz1/Ww1	: Z axis end point coordinate of thread part. --- Absolute/Incremental position
Ri	: Taper height constituent in thread part (radius value). When i = 0 is set, straight screw is made.
Pk	: Thread height. Designate the thread height in a positive radius value.
Q Δ d	: Cut depth. Designate the first cut depth in a positive radius value.
FI	: Thread lead

Configuration of one cycle

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.



- (a) First time Δd
- (b) Second time $\Delta d \times \sqrt{2}$
- (c) nth time $\Delta d \times \sqrt{n}$
- (d) Finishing allowance (Cut "m" times at finishing)

12.1.3.6 Compound Type Fixed Cycle for Turning Machining (Type II)

M system	—
L system	○

Pocket shapes can be machined in the longitudinal rough cutting cycle (G71) and face rough cutting cycle (G72).

The cutting method differs according to whether pocket machining is ON or OFF.

Pocket machining OFF Method to pull up the tool in a 45-degree direction from the workpiece

Pocket machining ON Method that traces the shape (After executing the last trace, the tool is pulled up in the X axis direction.)

Pocket machining is designated with the program (H address) or parameter.

Command format (This is a command format when the G71 is commanded. The G72 command is based on the G71 command.)

G71 Ud Re Hh ; <- (can be omitted when values set in parameters are used) G71 Aa Pp Qq Uu Ww Ff Ss Tt ;	
<H0:Used for finished shapes without pockets>	<H1:Mainly used for finished shapes with pockets>
<p>(a) Rough cutting start point</p>	<p>(a) Rough cutting start point (b) Hole bottom</p>
<p>Ud : Cut amount (modal) Reversible parameter Increment : μm or 1/10000inch Radius value command</p> <p>Re : Retract amount (modal) Reversible parameter Increment : μm or 1/10000inch Radius value command</p> <p>Hh : Pocket machining (modal) Reversible parameter 0 : Select this only for finished shapes without hollow areas (pockets). With the beginning of the pockets, the tool is pulled up in the 45-degree direction with each cycle until the finished shape is finally traced. 1 : This can be selected regardless of whether the finished shape has hollow (pocket) parts or not. A method that traces the finished shape with each cycle is used for the beginning of the pockets. Depending on the parameter setting, pocket machining ON/OFF is automatically determined by the number of axes in the finish shape start block.</p> <p>Aa : Finish shape program No. (If omitted, the program being executed is designated.) If the A command is omitted, the program being executed are applied. If A is omitted, the program following the end of this cycle will be executed at the block after Qq (finish shape end sequence No.). A file name can be designated instead of address A by enclosing the file name in brackets <>. (The file name can have up to 32 characters, including the extension.)</p> <p>Pp : Finish shape start sequence number (Head of program if omitted.)</p> <p>Qq : Finish shape end sequence number (To end of program if omitted.) If M99 precedes the Q command, up to M99.</p> <p>Uu : Finishing allowance in X axis direction (If omitted, finishing allowance in X axis direction is handled as 0.) Increment : μm or 1/10000inch Diameter/radius value command follows changeover parameter.</p> <p>Ww : Finishing allowance in Z axis direction (If omitted, finishing allowance in Z axis direction is handled as 0.) Increment : μm or 1/10000inch Radius value command</p> <p>Ff : Cutting feed rate (If omitted, cutting feed rate (modal) before G73 is applied.)</p> <p>Ss, Tt : Spindle command, tool command</p>	

12.1.4 Mirror Image

12.1.4.3 Mirror Image by G Code

M system	O
L system	—

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.

Gcode	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
```

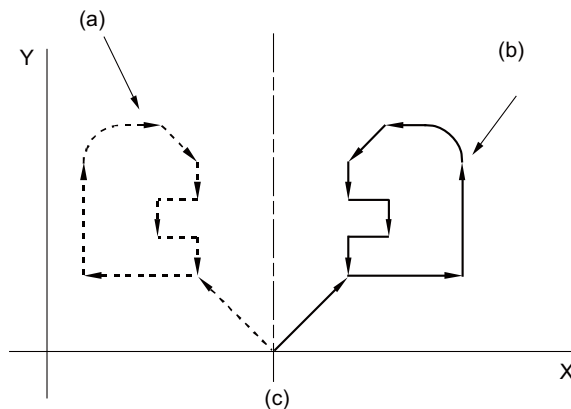
With the local coordinate system, the mirror image is applied with the mirror positioned respectively at x1, y1 and z1.

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 ;



- (a) Original shape (program)
- (b) Shape achieved when machining program for the left side has been executed after the mirror command
- (c) Mirroring axis

12.1.5 Coordinate System Operation

12.1.5.1 Coordinate Rotation by Program

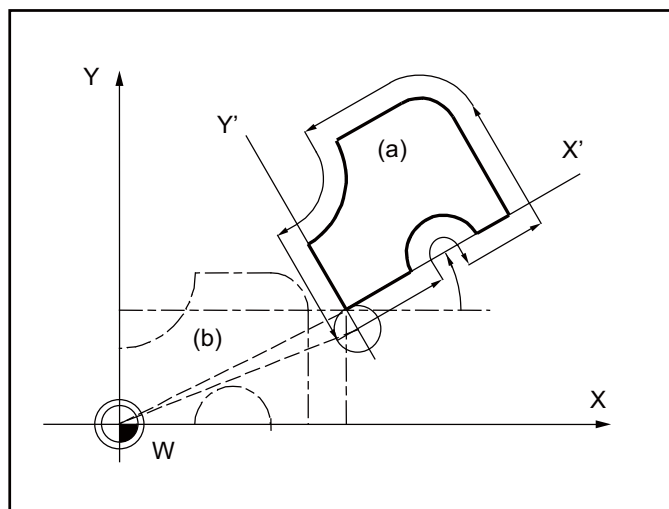
M system	○
L system	—

When it is necessary to machine a complicated shape at a position that has been rotated with respect to the coordinate system, you can machine a rotated shape by programming the shape prior to rotation on the local coordinate system, then specifying the parallel shift amount and rotation angle by means of this coordinate rotation command. The program format for the coordinate rotation command is given below.

(1) M system

G68	Xx1 Yy1 Rr1 ; Coordinate rotation ON
G69	Coordinate rotation cancel
G68	: Call command
G69	: Cancel command
Xx1,Yy1	: Rotation center coordinates
Rr1	: Angle of rotation

(Example)



(a) Actual machining shape
 (b) (Programmed coordinate)

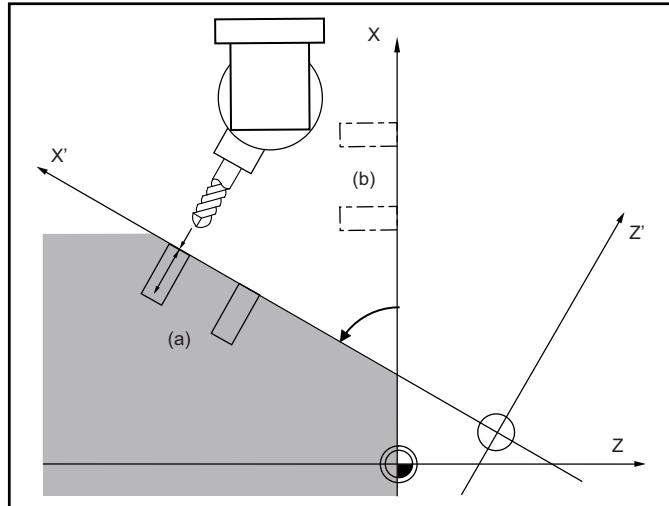
(2) L system

G68.1 Xx1 Yy1 Rr1 ; Coordinate rotation ON

G69.1 ; Coordinate rotation cancel

G68.1 :Call instruction
Xx1,Zz1 :Rotation center coordinates
Rr1 :Angle of rotation

(Example)



(a) Actual machining shape
(b) (Programmed coordinate)

12.1.6 Dimension Input

12.1.6.1 Corner Chamfering/Corner R

M system	<input type="radio"/>
L system	<input type="radio"/>

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.

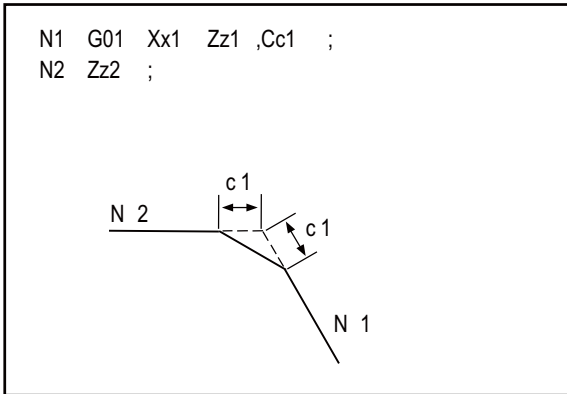
(1) Corner chamfering I/ Corner R I

When ",C" or ",R" is commanded for linear interpolation, corner chamfering or corner R can be inserted between linear blocks.

Corner chamfering I

Example:

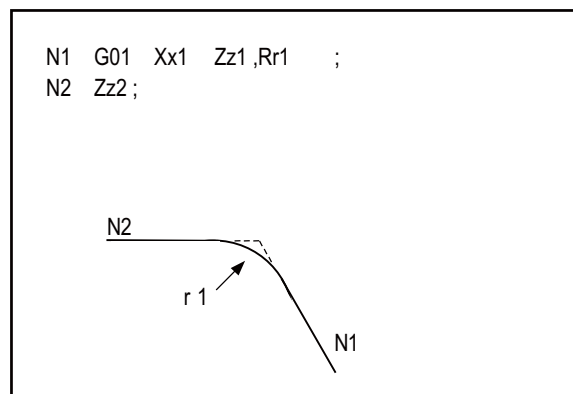
```
N1 G01 Xx1 Zz1 ,Cc1 ;
N2 Zz2 ;
```



Corner R I

Example:

```
N1 G01 Xx1 Zz1 ,Rr1 ;
N2 Zz2 ;
```



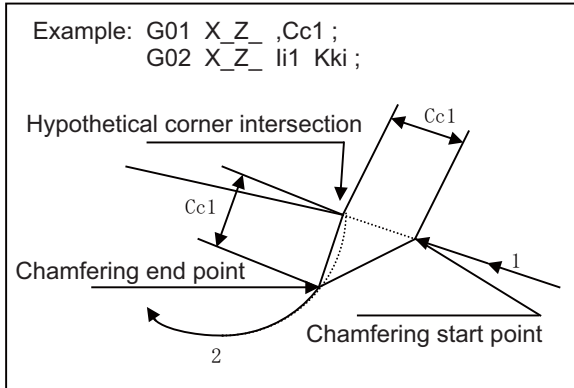
(Note 1) If a corner chamfering or corner R command is issued specifying a length longer than the N1 or N2 block, a program error occurs.

(2) Corner chamfering II/ corner R II

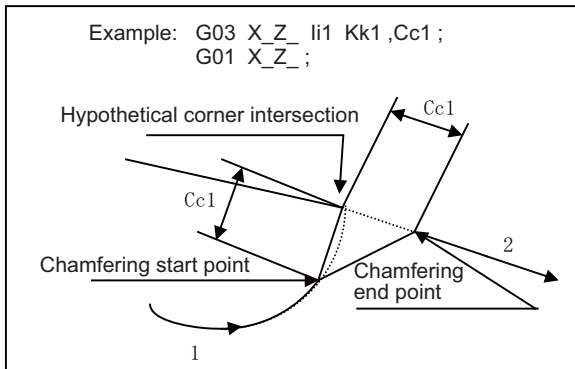
When ",C" or ",R" is commanded in a program between linear-circular, corner chamfering or corner R can be inserted between blocks.

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)

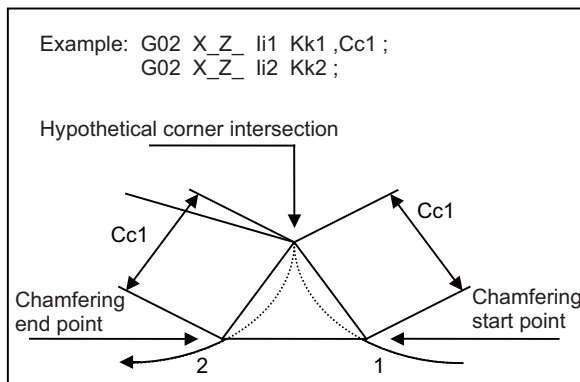
(a) Corner chamfering II (Linear - circular)



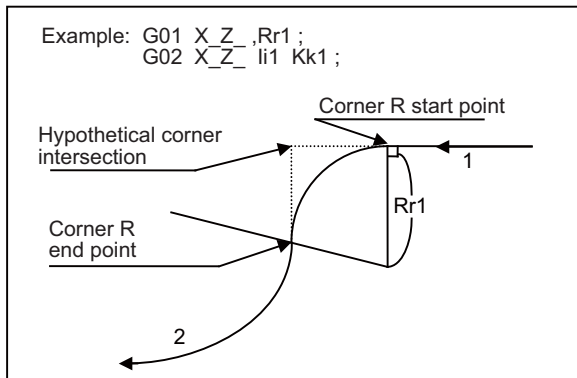
(b) Corner chamfering II (Circular - linear)



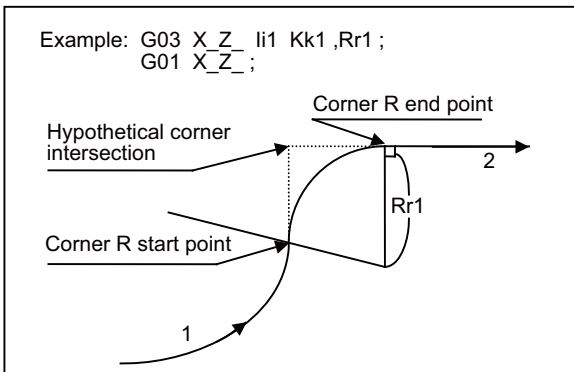
(c) Corner chamfering II (Circular - circular)



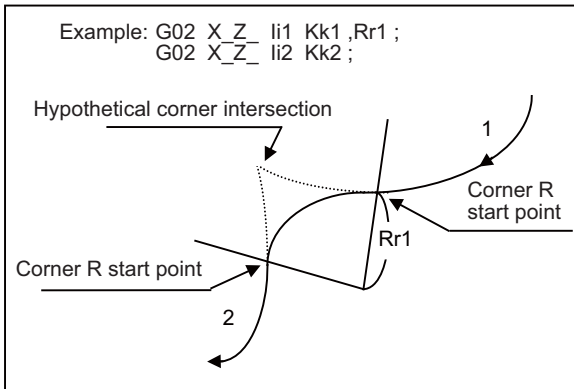
(d) Corner R II (Linear - circular)



(e) Corner R II (Circular - linear)



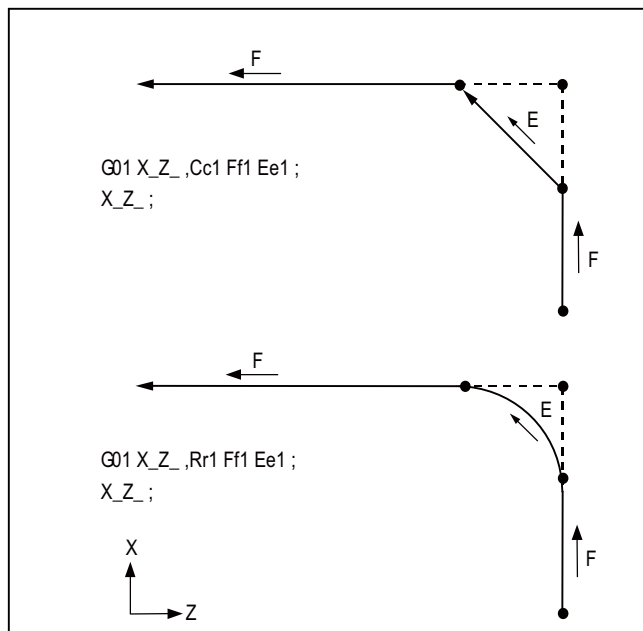
(f) Corner R II (Circular - circular)



(3) 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 and 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.)

(4) 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.

(a) Corner chamfering (I, K designation)

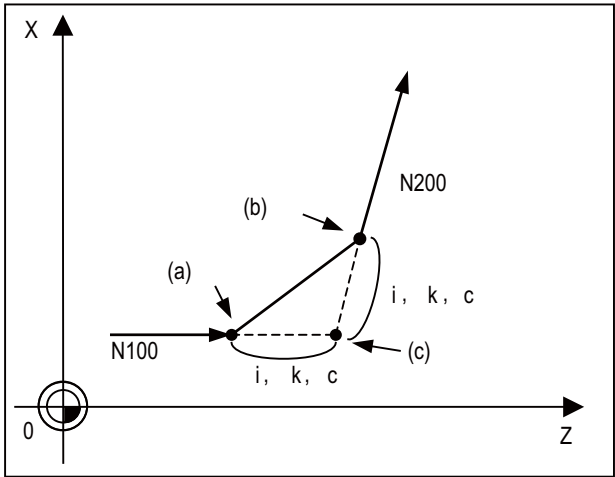
Corners are chamfered using the "I_", "K_" or "C_" address with no comma.

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.
```

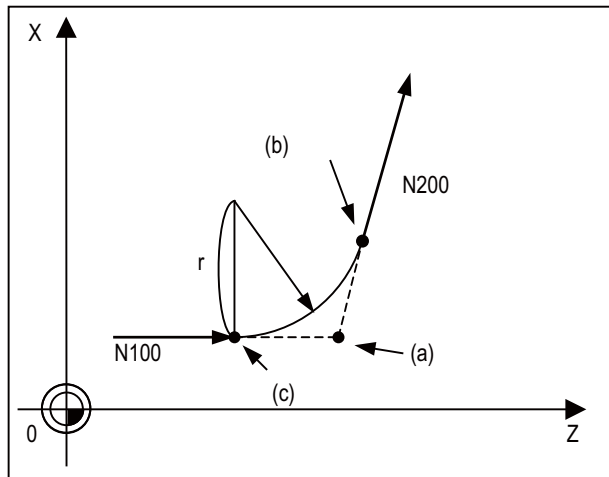


- (a) Chamfering start point
- (b) Chamfering end point
- (c) Hypothetical corner intersection

- 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 command 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.

- (b) Corner R (I, K designation)
 Corners are rounded using the "R_" address with no 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
```



- (a) Hypothetical corner intersection
- (b) Corner R end point
- (c) Corner R start point

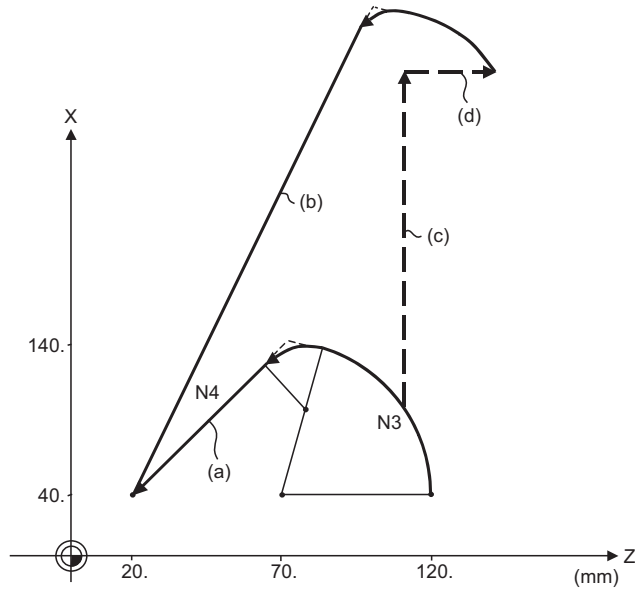
- If both corner chamfering and corner R commands are present in the same block, the last address will take effect.
- Corner R commands using the "R" address cannot be designated in an arc command block. "R" is regarded as the arc radius command in such a block.

(5) Interrupt during corner chamfering / Interrupt during corner R

Shown below are the operations of manual interruption during corner chamfering or corner R.

(a) With an absolute value command and manual absolute switch ON:

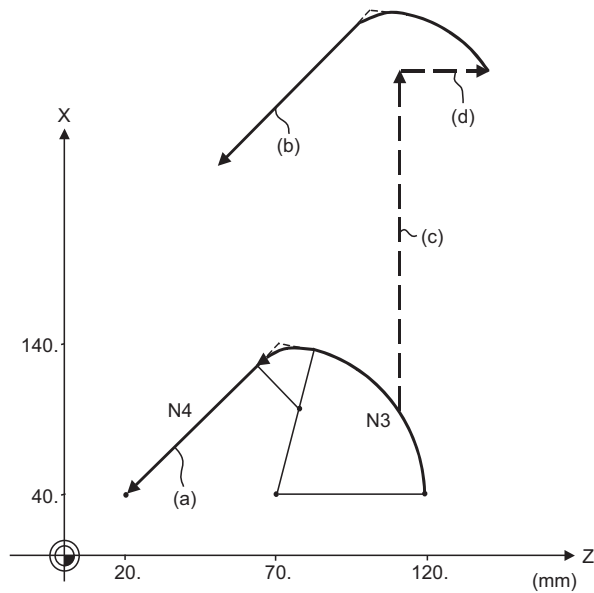
```
N1 G28 XZ;
N2 G00 X40. Z120.;
N3 G03 X140.Z70. K-50. ,R20. F100 ;
N4 G01 X40. Z20. ;
:
```



- (a) When interrupt is not applied
- (b) When interrupt is applied
- (c) X-axis interrupt
- (d) Z-axis interrupt

(b) With an incremental value 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. ;
:
```



- (a) When interrupt is not applied
- (b) When interrupt is applied
- (c) X-axis interrupt
- (d) Z-axis interrupt

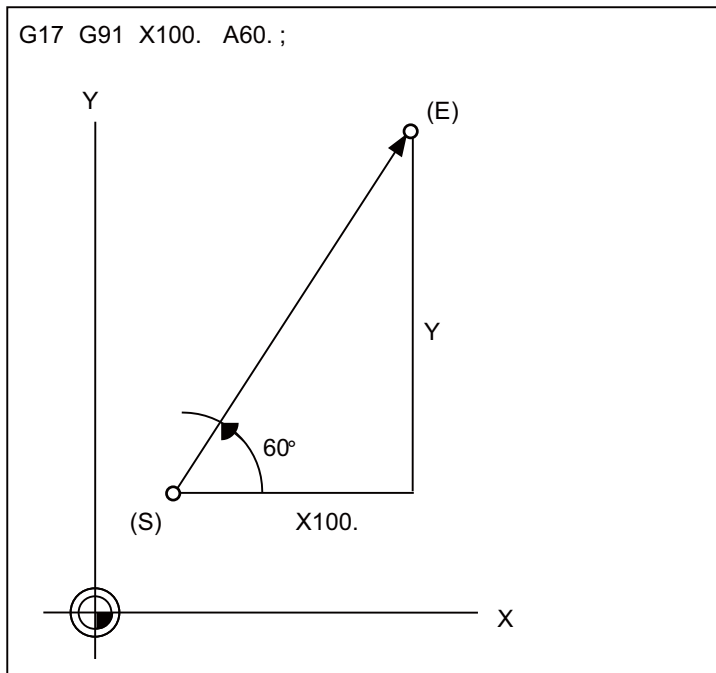
12.1.6.2 Linear Angle Command

M system	○
L system	○

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 ; or G17 Yy1 Aa1 ;
 G17 : Plane selection
 Xx1,Yy1 : 1 element of the end point coordinate
 Aa1 : Angle

Example



(E) End point (Automatically calculated with operation)
 (S) Start point

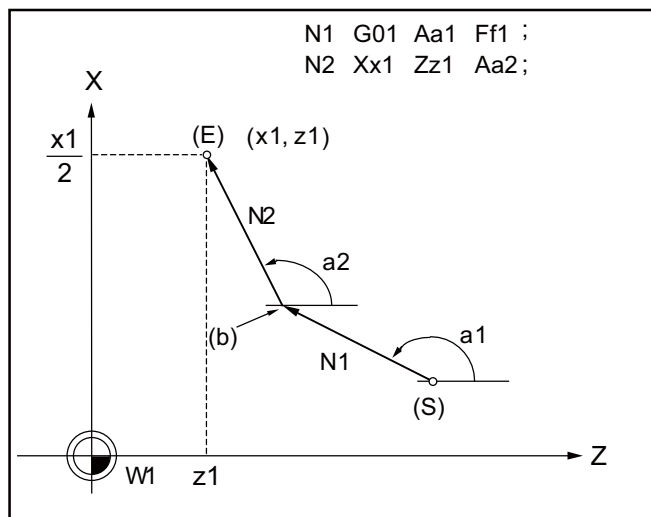
(Note 1) If the axis "A" or 2nd miscellaneous function "A" is used, address "A" is treated as the axis "A" command or the 2nd miscellaneous function, respectively.

12.1.6.3 Geometric Command

M system	O
L system	O

When it is difficult to find the intersection point of two straight lines with a continuous linear interpolation command, this point can be calculated automatically by programming the command for the angle of the straight lines.

Example



a: Angle (°) formed between straight line and horizontal axis on plane.
The plane is the selected plane at this time.

- (a) Automatic intersection point calculation
- (E) End point
- (S) Start point

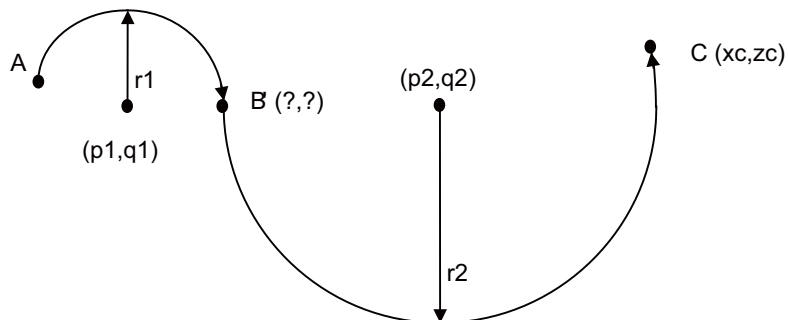
(Note1) This function cannot be used when using the A axis or 2nd miscellaneous function A.

(1) Automatic calculation of two-arc contact

When two continuous circular arcs contact with each other and it is difficult to find the contact, the contact is automatically calculated by specifying the center coordinates position or radius of the first circular arc and the end point (absolute position) and center position or radius of the second circular arc.

Example

```
G18 G02 Ii1 Kk1 Ff1 ;
G03 Xxc Zzc Ii2 Kk2 Ff2 ;
OR
G18 G02 Ii1 Kk1 Ff1 ;
G03 Xxc Zzc Rr2 Ff2 ;
OR
G18 G02 Rr1 Ff1 ;
G03 Xxc Zzc Ii2 Kk2 Ff2 ;
```



I and K : Incremental position from arc end point
P and Q : Arc center position (absolute position)

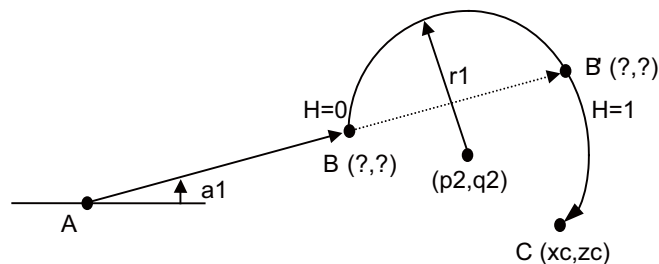
I and K are the arc center position (incremental position); distances from the start point in the first block or distances from the end point in the second block. P and Q (X, Z arc center position (absolute position)) can be commanded instead of I and K commands.

(2) Automatic calculation of linear-arc intersection

When it is difficult to find the intersections of a given line and circular arc, the intersections are automatically calculated by programming the following blocks.

Example

```
G18 G01 Aa1 Ff1 ;
G02 Xxc Zzc Ii2 Kk2 Hh2 (,Hh2) Ff2 ;
```



I and K : Incremental position from arc end point
P and Q : Arc center position (absolute position)
H = 0 : Intersection with shorter line (B point)
H = 1 : Intersection with longer line (B' point)

The p2 and q2 can be commanded instead of Ii2 and Kk2.

The linear - arc contact is automatically calculated by designating R instead of I and K (P, Q).

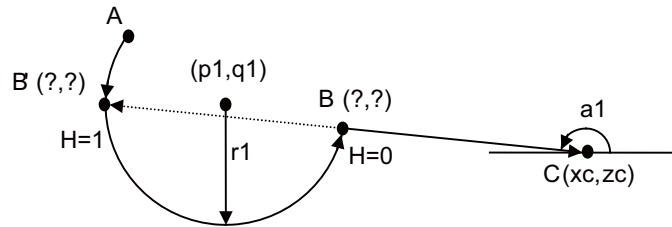
(3) Automatic calculation of arc-linear intersection

When it is difficult to find the intersections of a given circular arc and line, the intersections are automatically calculated by programming the following blocks.

Example

G18 G03 Ii1 Kk1 Hh1 Ff1 ;

G01 Xxc Zzc Aa1 Ff1 ;



- I and K : Incremental position from arc end point
- P and Q : Arc center position (absolute position)
- H = 0 : Intersection with shorter line (B point)
- H = 1 : Intersection with longer line (B' point)

The p1 and q1 can be commanded instead of Ii1 and Kk1.

The arc - linear contact is automatically calculated by designating R instead of I and K (P, Q).

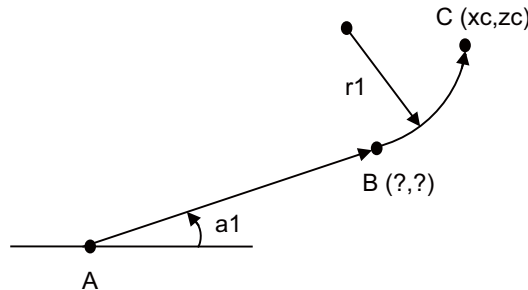
(4) Automatic calculation of linear-arc contact

When it is difficult to find the contact of a given line and circular arc, the contact is automatically calculated by programming the following blocks.

Example

G01 Aa1 Ff1 ;

G03 Xxc Zzc Rr1 Ff1 ;



The linear - arc intersection is automatically calculated by designating R instead of P and Q (I, K).

(5) Automatic calculation of arc-linear contact

When it is difficult to find the contact of a given circular arc and line, the contact is automatically calculated by programming the following blocks.

Example

G02 Rr1 Ff1 ;
G01 Xxc Zzc Aa1 Ff1 ;

The diagram shows a circular arc starting at point A and ending at point C(xc,zc). The arc's center is at (p1,q1) with radius r1. A line segment connects A and C, forming an angle a1 with the horizontal. The arc is tangent to the line segment at point B(?,?).

The arc - linear intersection is automatically calculated by designating R instead of P and Q (I, K).

12.1.7 Axis Control

12.1.7.3 Circular Cutting

M system	○
L system	—

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.

The program format is given below.

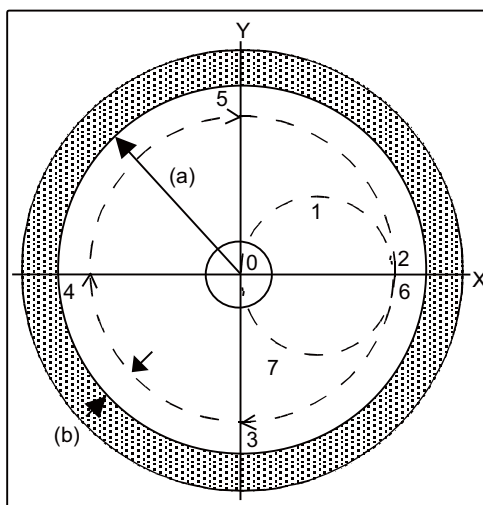
G12(G13) li Dd Ff ;	
G12	: Circular cutting command CW (clockwise)
G13	: Circular cutting command CCW (counter clockwise)
li	: Radius of complete circle
Dd	: Compensation No.
Ff	: Feed rate

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



- (a) Radius of circle
- (b) Compensation amount

(Note1) Circular cutting is carried out on the plane which has been currently selected (G17, G18 or G19).

(Note2) The (+) and (-) signs for the compensation amount denote reduction and expansion respectively.

12.1.9 Data Input by Program

12.1.9.1 Parameter Input by Program

M system	○
L system	○

The parameters set from the setting and display unit can be changed in the machining programs.
The data format used for the data setting is as follows.

```

G10 L70 ; Data setting start command
Pparameter No. Spart system No. Aaxis No. H □ data ; ..... Bit parameter;
Pparameter No. Spart system No. Aaxis No. Ddata ; ..... Numerical value parameter
Pparameter No. Spart system No. Aaxis No. <character string> ; ... Character string parameter

G11          Data setting end command
    
```

- (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.
- (Note 2) The part system No. is set to "1". It's allowed to omit address "S".
- (Note 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.
- (Note 4) Address H is commanded with the combination of setting data (0 or 1) and the bit designation (□) (0 to 7).
- (Note 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.
- (Note 6) The character string must be put in angled brackets "<" and ">".
If these brackets are not provided, the program error (P33) will occur.
Up to 63 characters can be set.
- (Note 7) Command G10L70, G11 in independent blocks. A program error (P33, P421) will occur if not commanded in independent blocks.
- (Note 8) The following data cannot be changed with the G10 L70 command:
Tool compensation data, workpiece coordinate data, PLC switch, and PLC axis parameter.
- (Note 9) 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

M system	○
L system	○

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.

(1) Workpiece coordinate system offset input

The position of the workpiece coordinate systems selected by the G54 to G59 commands can be set or changed by program commands.

Gcode	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)

The format for the workpiece coordinate system setting commands is shown below.

G10 L2(L20) Pp1 Xx1 Yy1 Zz1 ;	
G10 L2(L20)	: Workpiece coordinate system setting command
Pp1	: Workpiece coordinate system No.
Xx1,Yy1,Zz1	: Setting values

(Note 1) L2 can be omitted. Omitting Pp1 results in a program error. [M system]

(Note 2) L20 is used to designate the extended workpiece coordinate system.

(2) 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 amounts

TypeII

Gcode	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

The tool compensation input format is as follows.

G10 L11 Pp1 Rr1 ;	
G10 L11	: Command for setting compensation amount
Pp1	: Compensation No.
Rr1	: Compensation amount

(Note) When L11 has been omitted, the same operation will be performed as when L10 is selected. Omitting Pp1 results in a program error.

[L system]

Gcode	Function
G10 L10	Tool length compensation amount
G10 L11	Tool wear compensation amount

The tool compensation input format is as follows.

G10 L10(L11) Pp1 Xx1 Zz1 Rr1 Qq1 ;	
G10 L10(L11)	: Command for setting compensation amount
Pp1	: Compensation No.
Xx1	: X axis compensation amount
Zz1	: Z axis compensation amount
Rr1	: Nose R compensation amount
Qq1	: Hypothetical tool nose point

(Note) When L11 has been omitted, the same operation will be performed as when L10 is selected. Omitting Pp1 results in a program error.

12.1.10 Machining Modal

12.1.10.1 Tapping Mode

M system	○
L system	○

When tapping mode commands are issued, the NC system is set to the following internal control modes required for tapping.

1. Cutting override is fixed at 100%.
2. Deceleration commands at joints between blocks are invalid.
3. Feed hold is invalid.
4. Single block is invalid.
5. "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)

The machine is in the cutting mode status when its power is turned ON.

12.1.10.2 Cutting Mode

M system	○
L system	○

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)

The machine is in the cutting mode status when its power is turned ON.

12.2 Machining Accuracy Support Functions

12.2.1 Automatic Corner Override

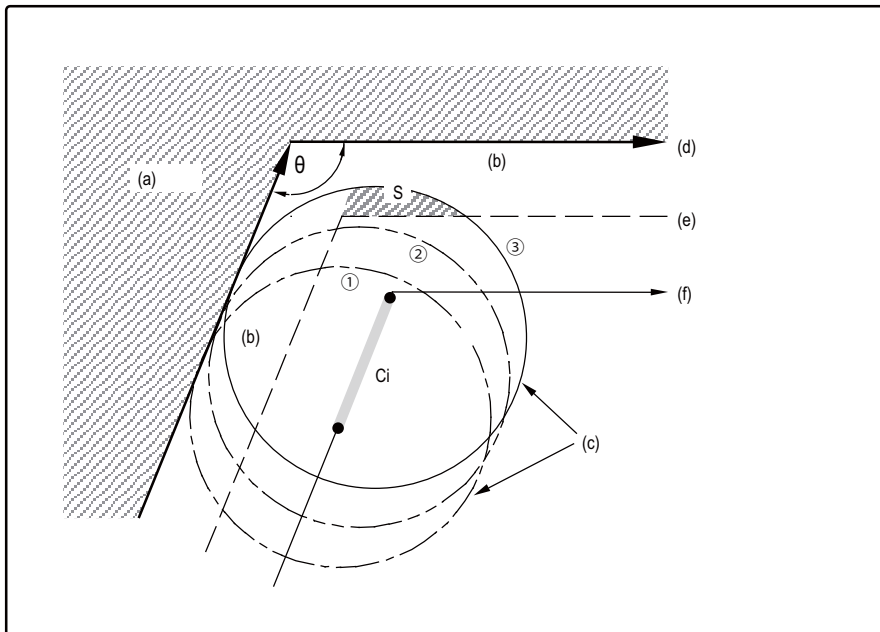
M system	○
L system	○

To prevent machining surface distortion due to the increase in the cutting load during cutting of corners, this function automatically applies an override on the cutting feed rate so that the cutting amount is not increased for a set time at the corner.

Automatic corner override is valid only during tool radius compensation.

The automatic corner override mode is set to ON by the G62 command and it is canceled by any of the G commands below.

- G40 Tool radius compensation cancel
- G61 Exact stop check mode
- G63 Tapping mode
- G64 Cutting mode



- (a) workpiece
- (b) Machining allowance
- (c) Tool
- (d) Programmed path (finished shape)
- (e) Workpiece surface shape
- (f) Tool center path
- θ : Max. angle at inside corner
- Ci : Deceleration range (IN)

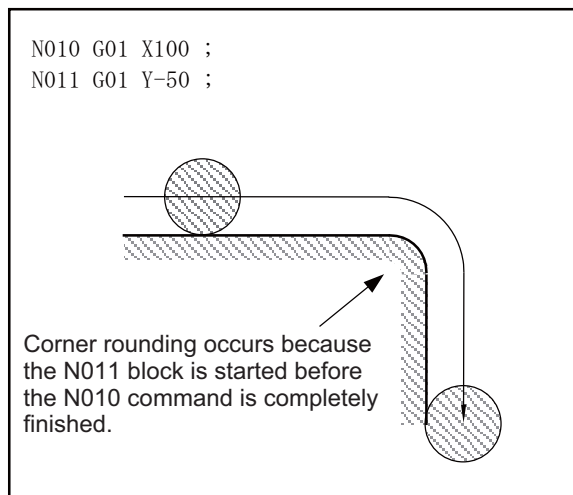
Operation

- (a) When automatic corner override is not to be applied :
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.
- (b) When automatic corner override is to be applied :
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.

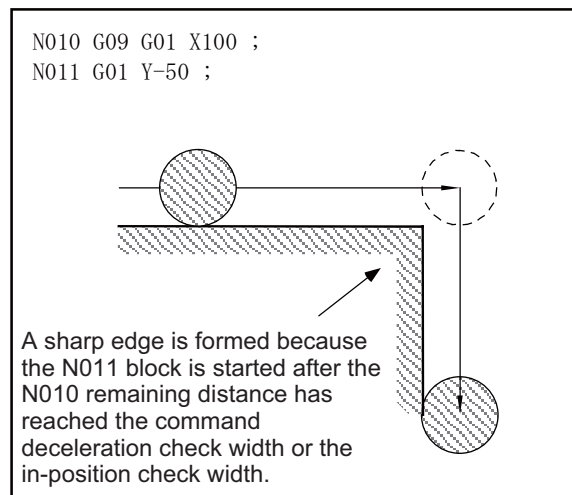
12.2.2 Deceleration Check

The deceleration check function leads the machine to decelerate and stop at the join between one block and another 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.

Without deceleration check



With deceleration check



The conditions for executing deceleration check are described below.

(1) 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.

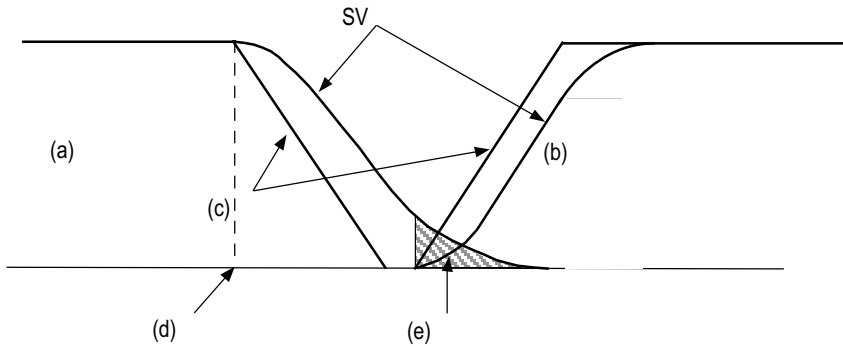
(2) Deceleration check in the cutting feed mode

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.

- (a) When G61 (exact stop check mode) is selected.
- (b) When the G09 (exact stop check) is issued in the same block.
- (c) When the error detect switch (external signal) is ON.

(3) Deceleration check system

Deceleration check is a system that executes the next block only after the command deceleration check is executed as shown below, and it has been confirmed that the position error amount, including the servo system, is less than the in-position check width (designated with parameter or with "I" in same block).



- SV: Servo
- (a) Previous block
- (b) Next block
- (c) Command
- (d) Block interpolation completion point
- (e) In-position check width

12.2.2.1 Exact Stop Check Mode

M system	○
L system	○

A deceleration check is performed when the G61 (exact stop check mode) command has been selected. G61 is a modal command. The modal command is released by the following commands.

- G62 Automatic corner override
- G63 Tapping mode
- G64 Cutting mode

Refer to "12.2.2 Deceleration Check" for details on the deceleration check.

12.2.2.2 Exact Stop Check

M system	○
L system	○

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 unmodal command.

Refer to "12.2.2 Deceleration Check" for details on the deceleration check.

12.2.2.3 Error Detection

M system	○
L system	○

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.

Refer to "12.2.2 Deceleration Check" for details on the deceleration check.

12.2.2.4 Programmable In-position Check

M system	○
L system	○

"," 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) command has been designated in the same block.
- When the G61 (exact stop check mode) command has been selected.

G01 Xx1 Zz1 Ff1 ,li1 ;	
Xx1, Zz1	: Linear interpolation coordinate position of each axis
Ff1	: Feed rate
,li1	: In-position width

As with linear interpolation, "," address is used to designate the in-position width for a positioning command from the machining program.

G00 Xx1 Zz1 ,li1 ;	
Xx1, Zz1	: Positioning coordinate position of each axis
,li1	: In-position width

In-position check operation

As for G01, after it is verified that the position error of the block in which the deceleration check is performed by the linear interpolation command (G01) is less than the in-position width of this command, the execution of the next block is commenced.

As for G00, after it is verified that the position error of the positioning command (G00: rapid traverse) is less than the in-position width of this command, the execution of the next block is commenced.

12.4 Programming Support Functions

12.4.1 Playback

M system	○
L system	—

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 Simple Programming

M system	○ (*11)
L system	○ (*11)

(*11) In case of FCU7-MU528-001

Create a part program by using NAVI MILL (for machining center system) or NAVI LATHE (for lathe system).

- (1) The following machining processes can be edited.

M system	L system
- Hole drilling (Drilling, deep hole drilling, step, boring, tapping) - Face 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 <Auxiliary operation> - Delivery

- (2) Cutting conditions are automatically determined from tool file and cutting condition file.
- (3) The operation screen consists of LIST VIEW area and OPERATION VIEW area. In the LIST VIEW area, the entire part program can be viewed at all time. In the OPERATION VIEW area, there are guide drawings related to each input item, which allows an easy data input.
- (4) 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.
- (5) Guidance function provides an operator with error recovery information.
- (6) Part program is a macro-program-based NC program. Process commands can be added in the edit screen.

12.4.4 G code Guidance

M system	○ (*2)
L system	○ (*2)

*2: Guidance data add-on is needed in case of FCU7-MU558-001.

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.

Machine Accuracy Compensation

13.1 Static Accuracy Compensation

13.1.1 Backlash Compensation

M system	○
L system	○

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). The amount of compensation for each axis ranges from 0 to ±9999 (pulses).

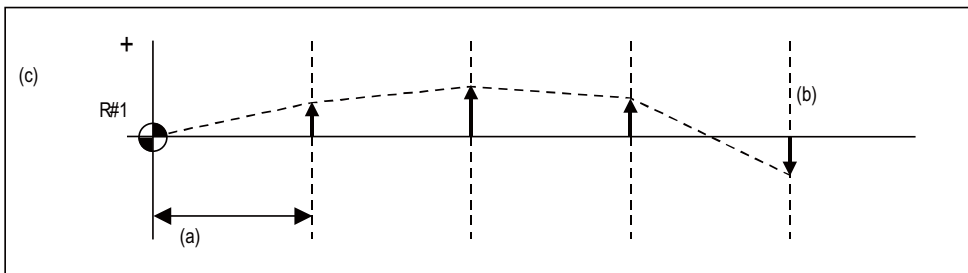
13.1.2 Memory-type Pitch Error Compensation

M system	○
L system	○

The machine accuracy can be improved by compensating for the errors in the screw pitch intervals among the mechanical errors (production errors, wear, etc.) of the feed screws. The compensation positions and amounts are stored in the memory by setting them beforehand for each axis, and this means that there is no need to attach dogs to the machine. The compensation points are divided into the desired equal intervals.

1. Division intervals of compensation points : 1 to 9999999 (μ m)
2. Number of compensation points : 1024
3. Compensation amount : -128 to 127 (output unit)
4. Number of compensation axes : 10 axes (including number of axes for memory-type relative position error compensation)

- (1) The compensation position is set for the compensation axis whose reference position serves as the zero (0) point. Thus, memory-type pitch error compensation is not performed upon the compensation axis for which return to reference position is not made after the controller power is turned ON and the servo is turned ON.
- (2) When the compensation axis is a rotary axis, select the dividing intervals so that one rotation can be divided.



- (a) Division interval
- (b) Compensation basic axis coordinate
- (c) Compensation amount

- (3) As shown in the figure above, highly individualized compensation control is exercised using the least output increment with linear approximation for the compensation intervals between the compensation points.

(Note 1) Compensation points 1,024 is a total including the points for memory-type relative position error compensation.
 (Note 2) A scale of 0 to 99-fold is applied on the compensation amount.

13.1.3 Memory-type Relative Position Error Compensation

M system	<input type="radio"/>
L system	<input type="radio"/>

Machine accuracy can be improved by compensating a relative error between machine axes, such as a production error or time aging.

The compensation base axis and compensation execution axis are set by using parameters.

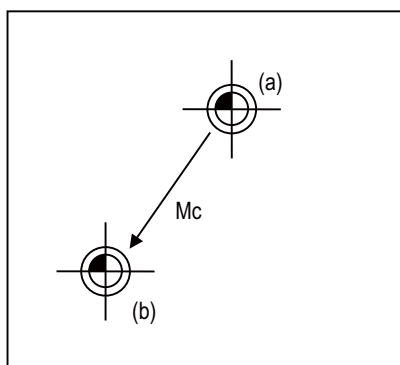
The compensation points are divided at any desired equal intervals.

1. Compensation point dividing intervals : 1 to 9999999 (μ m)
 2. Number of compensation points : 1024
 3. Compensation amount : -128 to 127 (output unit)
 4. Number of compensation axes : 10 axes (including number of axes for pitch error compensation.)
 - (1) The compensation position is set for the compensation axis whose reference position serves as the zero (0) point. Thus, memory-type relative position error compensation is not performed upon the axis for which return to reference position is not made after the controller power is turned ON and the servo is turned ON.
 - (2) When the compensation base axis is a rotary axis, select the dividing intervals so that one rotation can be divided.
 - (3) Since all coordinate systems of compensation execution axes are shifted or displaced by the compensation amount when the relative position error compensation is made, the stroke check point and machine coordinate system are also shifted or displaced.
- (Note 1) Compensation points 1,024 is a total including the points for memory-type pitch error compensation.
 (Note 2) A scale of 0 to 99-fold is applied on the compensation amount.

13.1.4 External Machine Coordinate System Compensation

M system	<input type="radio"/>
L system	<input type="radio"/>

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.



Mc: Compensation vector according to external machine coordinate system compensation

(a) Machine coordinate zero point when the external machine coordinate system offset amount is 0.

(b) Machine coordinate zero point

13.2 Dynamic Accuracy Compensation

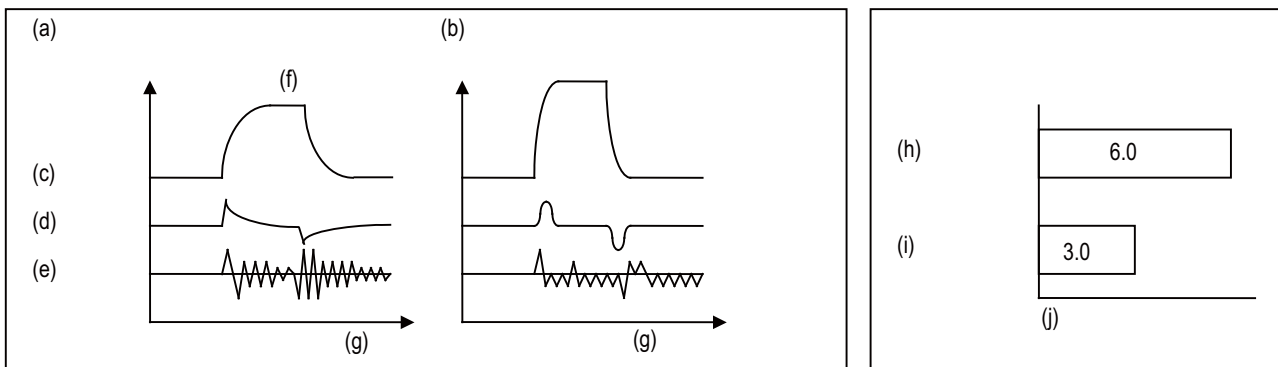
13.2.1 Smooth High-gain (SHG) Control

M system	○
L system	○

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.

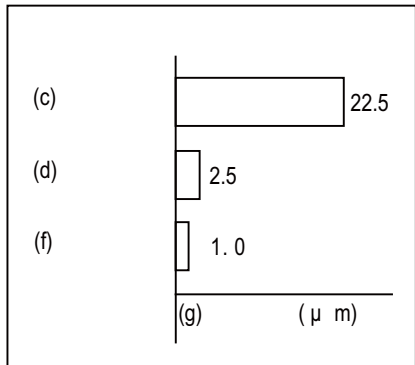
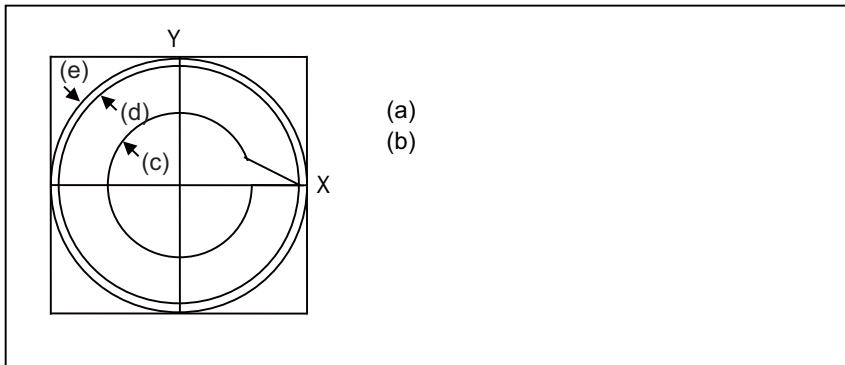
The features of the SHG control are as follows.

- (1) The acceleration/deceleration becomes smoother, and the mechanical vibration can be suppressed (approx. 1/2) during acceleration/deceleration. (In other words, the acceleration/deceleration time constant can be shortened.)



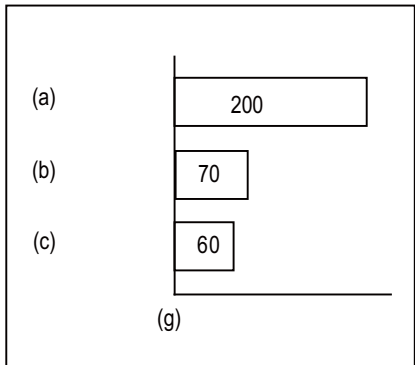
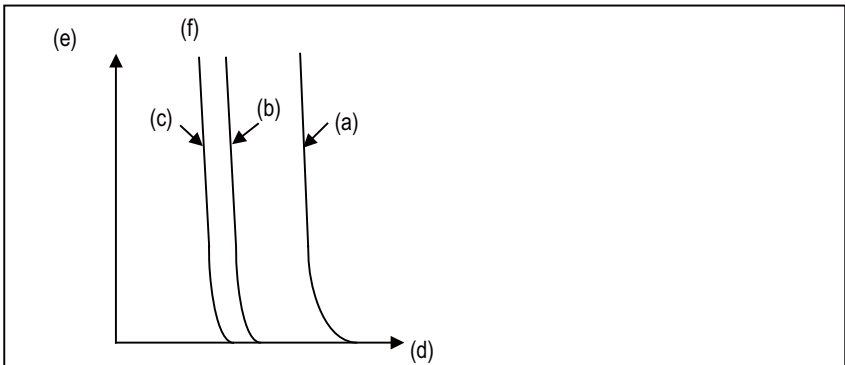
- (a) Conventional control (position loop gain = 33rad/S)
- (b) SHG control (position loop gain = 50rad/S)
- (c) Speed
- (d) Current
- (e) Machine vibration
- (f) Step response
- (g) Time
- (h) Conventional control
- (i) SHG control
- (j) Machine vibration amount (μ m)

(2) The shape error is approx. 1/9 of the conventional control.



- (a) Feed rate 3000mm/min.
- (b) Radius 50mm
- (c) Conventional control
- (d) SHG control
- (e) SHG control + FF (Feed forward)
- (f) SHG control + FF
- (g) Roundness error (μm)

(3) The positioning time is approx. 1/3 of the conventional control.



- (a) Conventional control
- (b) SHG control
- (c) SHG control + FF (Feed forward)
- (d) Time
- (e) Droop
- (f) Droop during rapid traverse deceleration
- (g) Positioning time (ms)

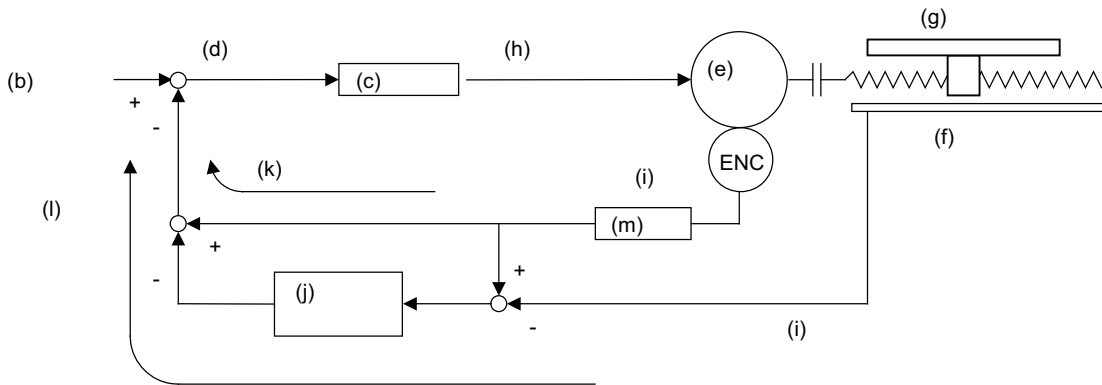
13.2.2 Dual Feedback

M system	<input type="radio"/>
L system	<input type="radio"/>

If the motor and machine coupling or machine system's rigidity is low (ex. large machine, etc.) when using a closed loop system, the response during acceleration/deceleration will vibrate and cause overshooting. This can cause the position loop gain from increasing. 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.



(a)

- (a) Dual feedback control
- (b) Position command
- (c) Position control
- (d) Position droop
- (e) Servo motor
- (f) Linear scale
- (g) Table
- (h) Speed command
- (i) Position FB
- (j) Primary delay filter
- (k) High frequency FB element
- (l) Low frequency FB element
- (m) dead band

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

M system	<input type="radio"/>
L system	<input type="radio"/>

This function compensates the error in the protrusion shape caused by lost motion at the arc quadrant changeover section during circular cutting.

Automation Support Functions

14.1 Measurement

14.1.1 Skip

14.1.1.1 Skip

M system	○
L system	○

When the external skip signal is input during linear interpolation with the G31 command, the machine feed is stopped immediately, the remaining distance is discarded and the commands in the next block are executed.

[M system]

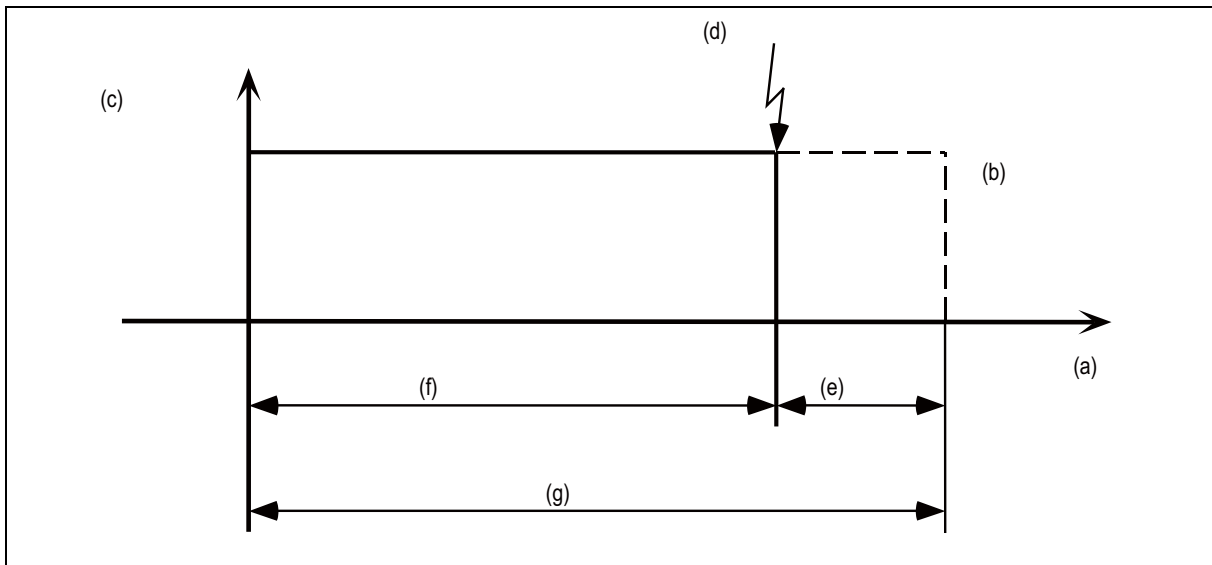
G31 Xx1 Yy1 Zz1 αα1 Rr1 Ff1 ; (α is the additional axis.)

G31 : Measurement command
 Xx1,Yy1,Zz1,αα1 : Command values
 Rr1 : Acceleration/deceleration command
 Ff1 : Feed rate

[L system]

G31 Xx1/Uu1 Zz1/Ww1 Rr1 Ff1 ;

G31 : Measurement command
 Xx1,Uu1,Zz1,Ww1 : Command values
 Rr1 : Acceleration/deceleration command
 Ff1 : Feed rate



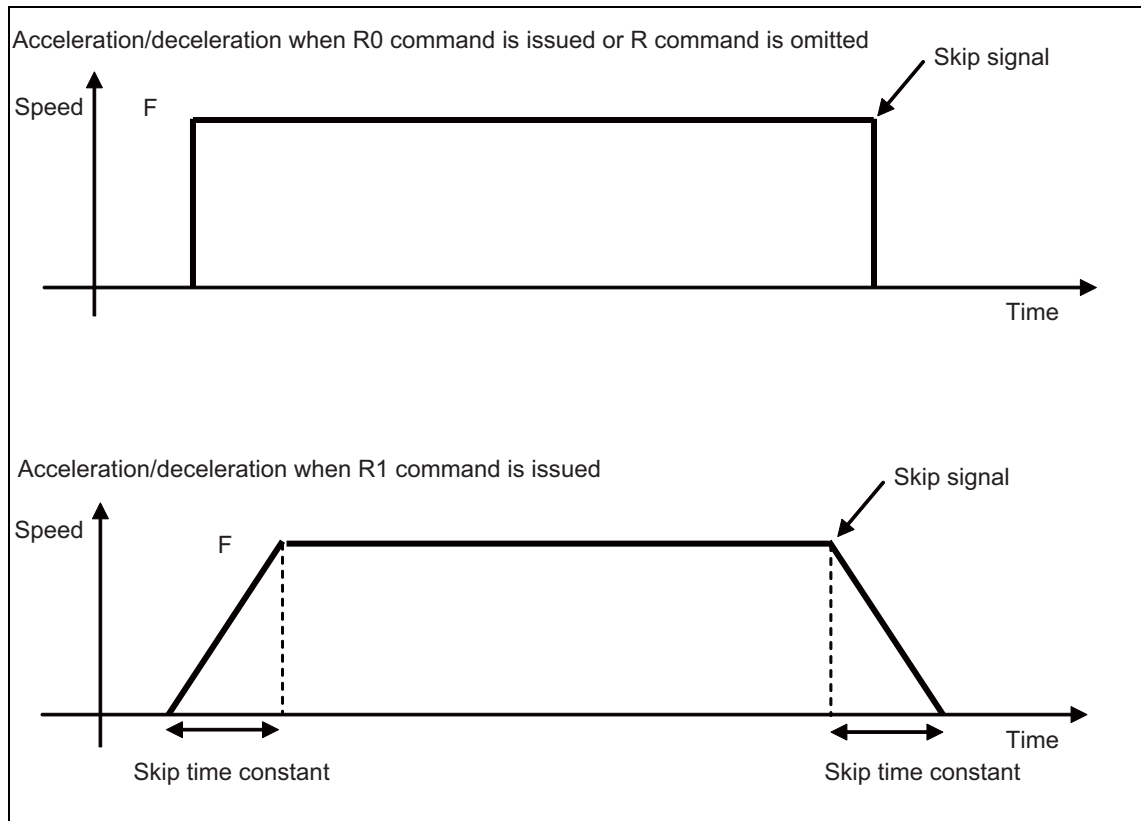
- (a) Position
- (b) Programmed end point
- (c) Feed rate
- (d) Skip signal input
- (e) Remaining distance
- (f) Actual movement distance
- (g) Command value

Command by G31 command basis for acceleration/deceleration command (R0/R1).

When R0 command is issued or R command is omitted, automatic acceleration/deceleration is not performed but step acceleration/deceleration is performed after interpolation in G31 block.

If R1 command is issued, automatic acceleration/deceleration is performed after interpolation even when skip signal is input.

A command is not issued or a command other than R0/R1 is issued, it will be the operation of acceleration/deceleration time constant=0(R0) and automatic acceleration/deceleration is not performed after interpolation.



Changeover of acceleration/deceleration type with R command

There are two types of skip feed rate.

- (1) Feed rate based on program command when F command is present in program
- (2) Feed rate based on parameter setting when F command is not present in program

(Note 1) The approximate coasting distance up to feed stop based on the detection delay in the skip signal input is calculated as below.

$$\delta \doteq \frac{F}{60} \times (T_p + t)$$

δ : Coasting distance (mm)

F : G31 rate (mm/min)

T_p : Position loop time constant (s) = (position loop gain)⁻¹

t : Response delay time of 0.0035 (s)

(Note 2) Skipping during machine lock is not valid.

14.1.1.2 Multiple-step Skip

M system	○
L system	○

This function realizes skipping by designating a combination of skip signals for each skip command.

(1) 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)

(2) 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)

- (a) Specify the skip rate in command feedrate F. However, F modal is not updated.
- (b) 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.
- (c) 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	○	○	○	○	○	○	○	○

14.1.2 Automatic Tool Length Measurement

M system	○
L system	○

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.

(1) 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.

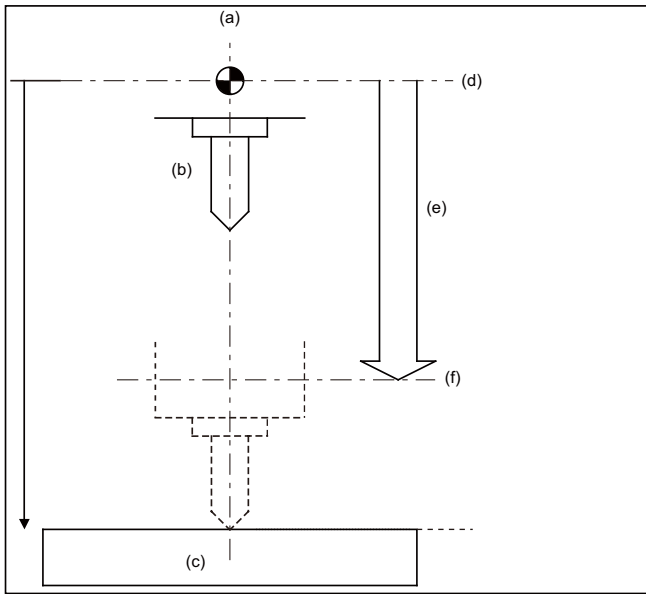
If compensation has already been applied to the tool, it is moved in the direction of the measurement position with the compensation still applied, and when the measurement and calculation results are such that a further compensation amount is to be provided, the current compensation amount is further 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.



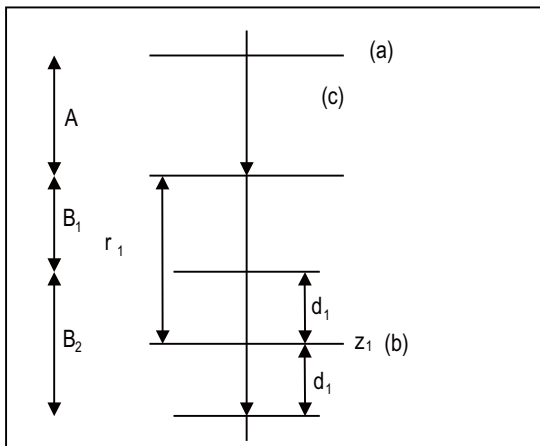
At this time, the tool length compensation amount has a minus ("-") value.

Example of program

```
G28 Z0 ;
T01 ;
M06 T02 ;
G43 G00 Z0 H01 ;
G37 Z-300. R10. D2. F10 ;
:
```

In this case, the distance ($H01 = Z_{a1} - z_0$) 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.

- (a) Tool change position
- (b) Tool
- (c) Sensor
- (d) Reference position (In case of machine coordinate system zero point.)
- (e) Amount of movement based on tool length measurement
- (f) Tool length measurement position (Z_{a1})



Area A : Moves with rapid traverse feed rate.
 Areas B₁, B₂ : Moves with the measurement speed (f1 or parameter setting)
 If a sensor signal is input in area B₁, an error will occur.
 If a sensor signal is not input in the area B₂, an error will occur.

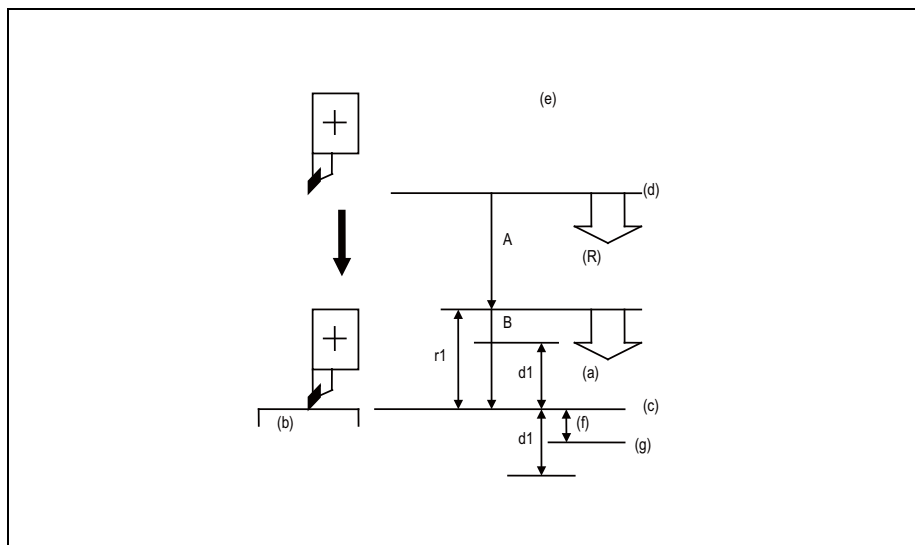
- (a) Start point
- (b) Measurement position
- (c) r₁, d₁, and f₁ can also be set in parameters.

(2) Automatic tool length measurement (L series)

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.	



- (a) F feed
- (b) Measuring instrument
- (c) Measurement position
- (d) Start position
- (e) r1, d1, and f1 can also be set in parameters.
- (f) Compensation amount
- (g) Sensor ON
- (R) Rapid traverse feed

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.

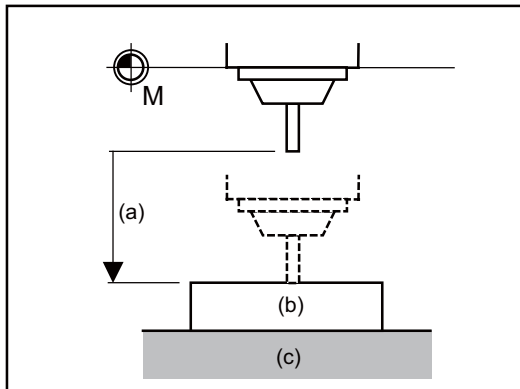
14.1.3 Manual Tool Length Measurement 1

M system	<input type="radio"/>
L system	<input type="radio"/>

Simple measurement of the tool length is done without a sensor.

(1) Manual tool length measurement I [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.



- (a) Manual movement amount (tool length compensation amount)
- (b) Workpiece
- (c) Table

(2) Manual tool length measurement I [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 two types of measurement methods in manual tool length measurement I: the basic point method and the measurement value input method. The method is selected by setting parameter. For key operation there are normal operation mode/simple operation mode.

(a) Basic point method

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

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

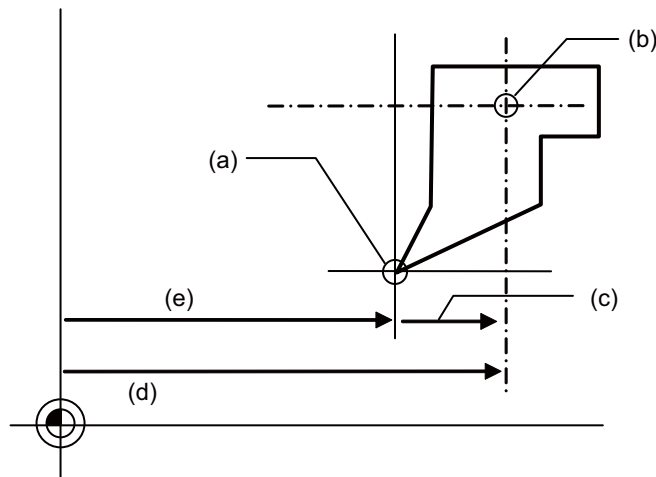
When measurement position is set to parameter or workpiece coordinate offset (modal).

<Parameter designation method>

The tool length is automatically calculated using the following formula.

$$\text{Tool length} = \text{Machine coordinates} - \text{Measurement point (\#2015 tml-)}$$

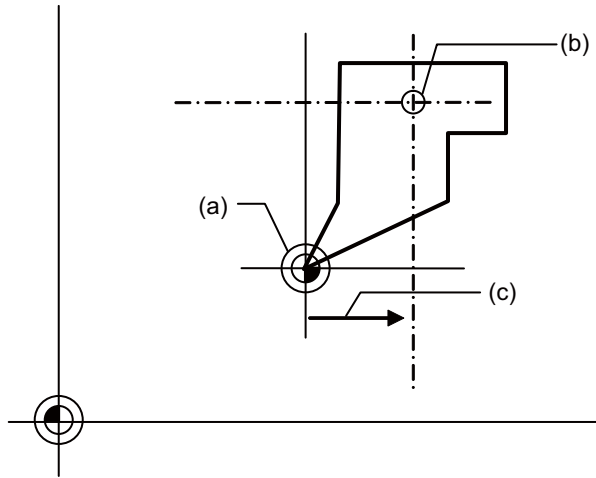
(Note) Always set the measurement point with a radius value, regardless of the diameter/radius command.



- (a) Measurement point
- (b) Tool length basic point
- (c) Z-axis tool length
- (d) Z-axis machine position
- (e) #2015 tml-

<Workpiece coordinate offset designation method>

Tool length is the coordinate value on the workpiece coordinate system.



- (a) Measurement position(Workpiece coordinate origin (modal))
- (b) Tool length basic point
- (c) Workpiece coordinate position = Z-axis tool length

(b) Measurement value input method

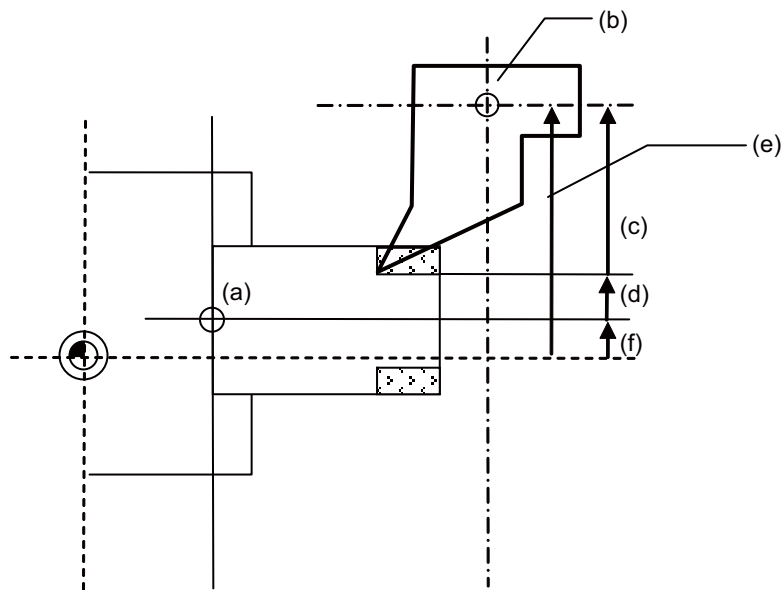
Actually cut the workpiece. Measure its dimensions, and obtain the tool length from the measured values. The measurement basic point is characteristic for each machine (the center of the chuck face, etc. designated by parameter), or workpiece coordinate offset (modal).

<Parameter designation method>

The tool length is automatically calculated by the equation below.

$$\text{Tool length} = \text{Machine coordinate values} - \text{Measurement basic point (\#2015 tml-)} - \text{Measured value}$$

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

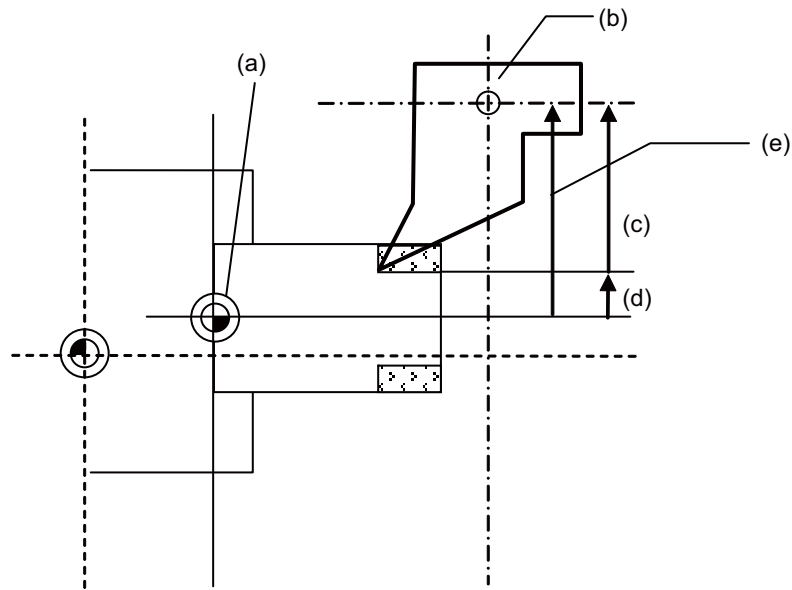


- (a) Measurement basic point
- (b) Tool length basic point
- (c) X-axis tool length
- (d) X-axis measured value
- (e) X-axis machine position
- (f) #2015 tml-

<Workpiece coordinate offset designation method>

The tool length is automatically calculated by the equation below.

$$\text{Tool length} = \text{Workpiece coordinate position} - \text{Measured value}$$



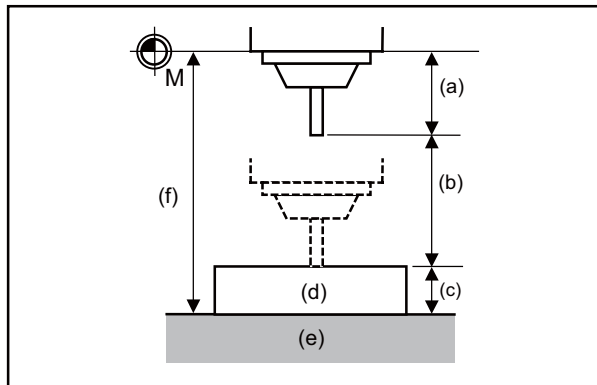
- (a) Measurement basic point (Wk. coordinate origin (modal))
- (b) Tool length basic point
- (c) X-axis tool length
- (d) X-axis measured value
- (e) X axis WK. coordinate position

14.1.4 Manual Tool Length Measurement 2

M system	<input type="radio"/>
L system	<input type="radio"/>

(1) Manual tool length measurement II [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.



- (a) Tool length compensation amount
- (b) Manual movement amount
- (c) Basic height
- (d) Gauge block
- (e) Table
- (f) TLM basic length (setup parameter)

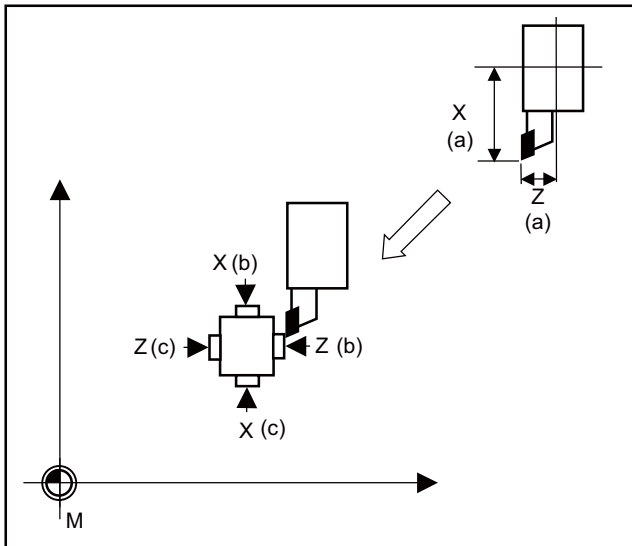
If the height axis designation parameter is ON, the axis designated for plane selection basic axis K is the axis targeted for measurement as the height axis.

Furthermore, if the tool length measurement check parameter is ON, an input OK/cancel confirmation message appears after input key has been pressed.

(2) Manual tool length measurement II [L system]

A device in which a touch sensor is built is used. Simply by causing the tool nose to touch the touch sensor in manual feed, the tool compensation amount can be calculated and stored in tool compensation amount memory.

Preset the machine coordinate position of the touch sensor touch face in parameter as the measurement basic position.



- (a) Tool length
- (b) - touch face
- (c) + touch face

14.2 Tool Life Management

14.2.1 Tool Life Management

In this type, how long and how many times the program commanded tool is used are accumulated to monitor the usage state.

14.2.1.1 Tool Life Management I

M system	<input type="radio"/>
L system	<input type="radio"/>

The following two method can be selected.

- Management by the time of use
 The cutting time after specification of a tool selection (T) command (G01, G02, and G33) is added to the tool use time for the specified tool.
- Management by the frequency of use
 The tool use counter corresponding to the specified tool No. is incremented each time a tool selection (T) command is specified for the tool.

(1) M system

The life of up to 1000 tools can be managed. The maximum number of tools to be registered varies depending on options.

When the number of uses has reached the limit, the "Tool life over" signal will turn ON.

(2) L system

The life for up to 80 tools (tool numbers 1 to 80) can be managed.

When a T command is issued after it has reached the limit, the "Tool life over" signal will turn ON.

14.2.1.2 Tool Life Management II

M system	<input type="radio"/>
L system	<input type="radio"/>

(1) 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 life is the longest of all in the group is selected.

- Number of groups: Max. 200 sets
- Number of tools in group: Max. 200 tools

(2) L system

The life of each tool (time and frequency) is controlled, and when the life is reached, a spare tool that is the same type is selected from the group where the tool belongs and used.

- Number of groups: Max. 80 sets
- Number of tools in group: Max. 16 tools

14.2.1.3 Tool Life Management III

M system	○
L system	—

For the user PLC designated tool, that tool's usage time (0 to 4000 hours) or frequency of use (0 to 65000 times) is accumulated, and the tool usage state is monitored. The life for up to 200 tools can be managed.

This function is not controlled by the group No.

14.2.2 Number of Tool Life Management Sets

14.2.2.1 80 Sets

M system	—
L system	○

14.2.2.2 200 Sets

M system	○
L system	—

14.3 Others

14.3.1 Programmable Current Limitation

M system	○
L system	○

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%

- (1) If the current limit is reached when the current limit is valid, the current limit reached signal is output.
- (2) 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.
- (3) 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.)
- (4) The setting range of the current limit value is 1% to 999%. Commands that exceed this range will cause a program error.
- (5) 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%.
- (6) 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".

Safety and Maintenance

15.1 Safety Switches

15.1.1 Emergency Stop

M system	○
L system	○

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 signal is turned OFF.

15.1.2 Data Protection Key

M system	○
L system	○

With the input from the user PLC, it is possible to prohibit the parameter setting or deletion, and the program edit on setting and display unit.

Data protection is divided into the following groups.

Group 1 : For protecting the tool data and protecting the coordinate system presettings as based on origin setting (zero)

Group 2 : For protecting the user parameters and common variables

Group 3 : For protecting the machining programs

15.2 Display for Ensuring Safety

15.2.1 NC Warning

M system	○
L system	○

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	Description
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

M system	○
L system	○

The alarms which are output by the NC system are listed below. When one of these alarms 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.

Type of alarm	Description
Operation alarm	This alarm occurring due to incorrect operation by the operator during NC operation and that by machine trouble are displayed.
Servo alarm	This alarm describes errors in the servo system such as the servo drive unit motor and encoder.
Spindle alarm	This alarm describes errors in the spindle system such as the spindle drive unit motor and encoder.
MCP alarm	An error has occurred in the drive unit and other interfaces.
System alarm	This alarm is displayed on the screen with the register at the time when the error occurred if the system stops due to a system error.
Absolute position detection system alarm	An alarm in the absolute position detection system is displayed.
Auxiliary axis alarm	The auxiliary axis alarm is displayed.
Computer link error	The computer link alarm is displayed.
User PLC alarm	The user PLC alarm is displayed.
Program error	This alarm occur during automatic operation, and the cause of this alarm is mainly program errors which occur, for instance, when mistakes have been made in the preparation of the machining programs or when programs which conform to the specification have not been prepared.
Network service error	The network service alarm is displayed.

15.2.3 Operation Stop Cause

M system	○
L system	○

The stop cause of automatic operation is displayed on the setting and display unit.

15.2.4 Emergency Stop Cause

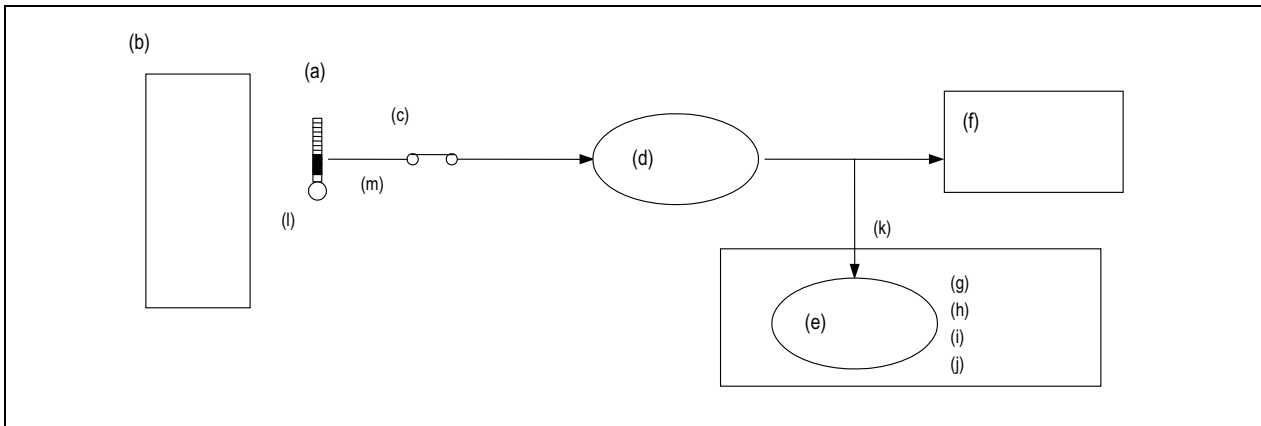
M system	○
L system	○

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

M system	<input type="radio"/>
L system	<input type="radio"/>

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.
 The overheat alarm occurs at 80 C or more in the control unit.



- (a) Overheat detection
- (b) Control unit
- (c) Parameter
- (d) Temperature alarm
- (e) User PLC
- (f) Message display
- (g) Cooling fan rotation
- (h) Lamp alarm
- (i) Emergency stop
- (j) Others
- (k) Bit device
- (l) (a temperature of 80 C)
- (m) (Default: valid)

(Note 1) If the parameter is used to set the temperature rise detection function to invalid, overheating may occur, thereby disabling control and possibly resulting in the axes running out of control, which in turn may result in machine damage and/or bodily injury or destruction of the unit. It is for this reason that the detection function is normally left "valid" for operation.

15.2.6 Battery Alarm/Warning

M system	<input type="radio"/>
L system	<input type="radio"/>

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.

15.3 Protection

15.3.1 Stroke End (Over Travel)

M system	○
L system	○

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 areas where tool entry is prohibited can be set.

The stored stroke limits I and II are handled as follows.

Type	Prohibited range	Explanation
I	Outside	- Set by the machine tool builder. - When used with II, the narrow range designated by the two types becomes the movement valid range.
II	Outside	- Set by the user. - By using the program command, the parameter can be changed and the function can be switched to ON/OFF.

15.3.2.1 Stored Stroke Limit I/II

M system	○
L system	○

(1) Stored Stroke Limit I

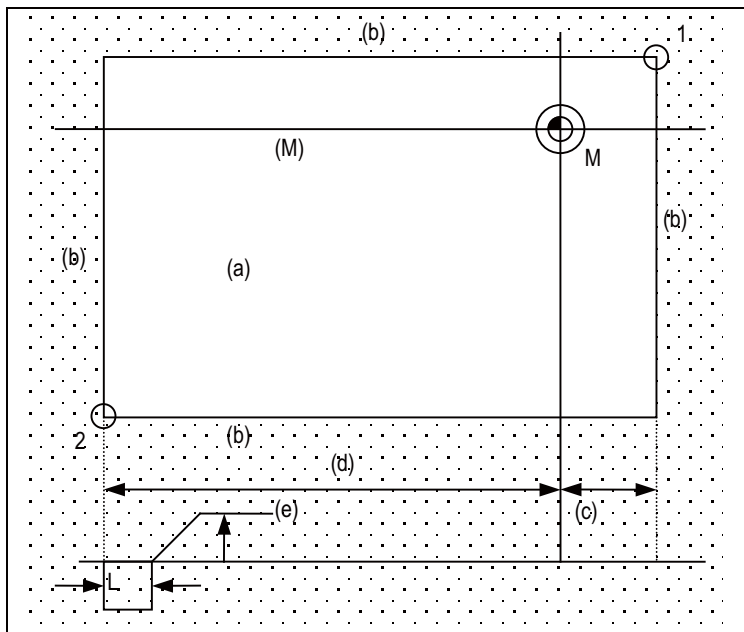
This is the stroke limit function used by the machine tool builder, and the area outside the set limits is the entrance prohibited area.

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 of both functions is the movement valid range.

The setting range is -99999.999 to +99999.999mm.

The stored stroke limit I function is made valid not immediately after the controller power is turned ON but after reference position return.

The stored stroke limit I function will be invalidated if the maximum and minimum values are set to the same data.



The position of points 1 and 2 are set using the machine coordinate.

: Prohibited area

- (a) Machine movement valid range
- (b) Prohibited area
- (c) "+" setting
- (d) "-" setting
- (e) Feedrate
- (M) Machine coordinate system
- 1: Point 1
- 2: Point 2

All axes will decelerate and stop if an alarm occurs even for a single axis during automatic operation. Only the axis for which the alarm occurs will decelerate and stop during manual operation. The stop position must be before the prohibited area.

The value of distance "L" between the stop position and prohibited area differs according to the feed rate and other factors.

The range 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 range 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 by with the R register values.

This feature allows an operator to switch the range of the stored stroke limit I during axis movement for the purpose of tool exchange, for example.

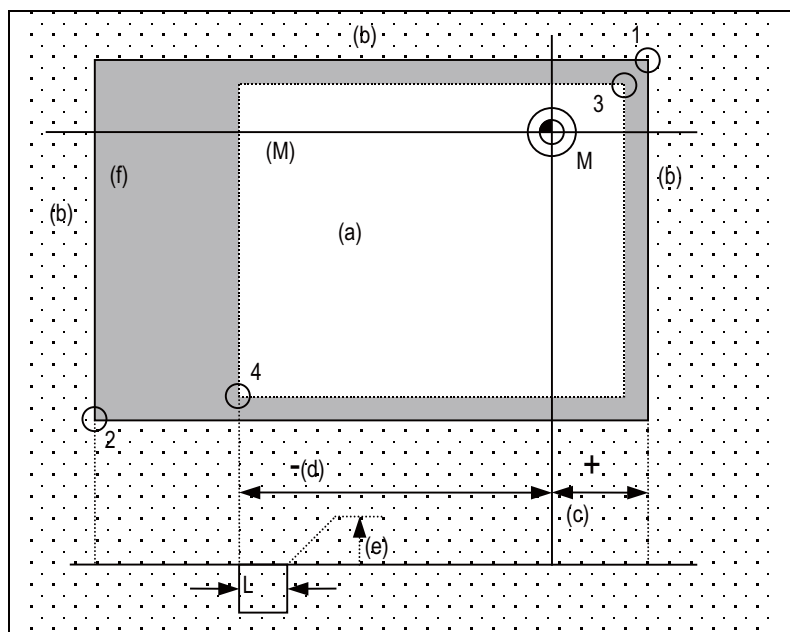
(2) Stored Stroke Limit II

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 maximum and minimum values for each axis can be set by parameters. The function itself is used together with the stored stroke limit I function described in the foregoing section, and the tolerable area of both functions is the movement valid range.

The setting range is -99999.999 to +99999.999mm.

The stored stroke limit II function will be invalidated if the maximum and minimum parameter values are set to the same data.



The position of points 3 and 4 are set with the machine coordinate.
The area determined by points 1 and 2 is the prohibited area set with stored stroke limit I.

  : Prohibited area

- (a) Machine movement valid range
- (b) Prohibited area
- (c) "+" setting
- (d) "-" setting
- (e) Feed rate
- (f) Area prohibited by stored stroke limit function II
- (M) Machine coordinate system
- 1: Point 1
- 2: Point 2
- 3: Point 3
- 4: Point 4

All axes will decelerate and stop if an alarm occurs even for a single axis during automatic operation. Only the axis for which the alarm occurs will decelerate and stop during manual operation. The stop position must be before the prohibited area.

The value of distance "L" between the stop position and prohibited area differs according to the feed rate and other factors.

The stored stroke limit II function can also be invalidated with the parameter settings.

15.3.4 Chuck/Tailstock Barrier Check

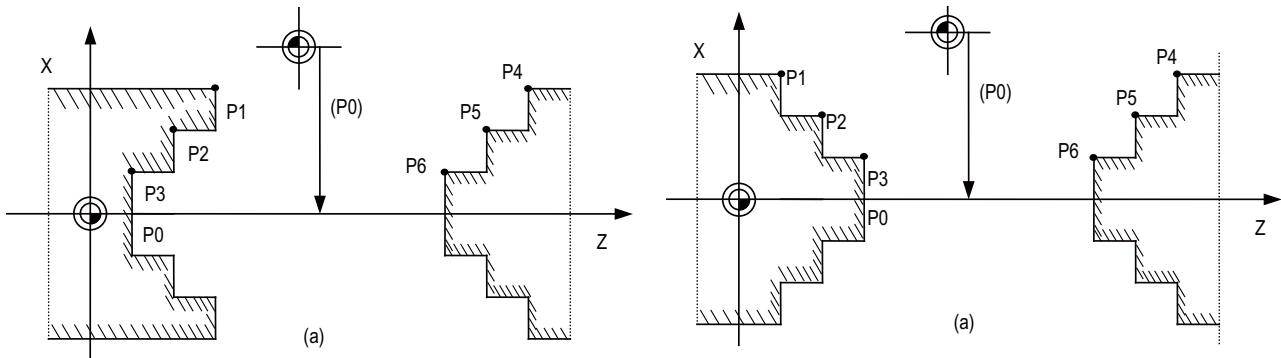
M system	—
L system	○

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.

Program format

```
G22 ; ..... Barrier ON
G23 ; ..... Barrier OFF (cancel)
```

- (1) When the machine is about to exceed the area, the machine is stopped and an alarm is displayed. To cancel the alarm, execute reset.
- (2) The function is also effective when the machine is locked.
- (3) This function is valid when all axes for which a barrier has been set have completed reference position return.
- (4) Chuck barrier/tailstock barrier setting.



(a) (Form 1)

(a) (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:

$$P1 \geq P2 \geq P3, P4 \geq P5 \geq P6$$

(However, this need not apply to the Z axis coordinates.)

15.3.5 Interlock

M system	○
L system	○

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.

- (1) In the manual mode, only that axis for which the interlock signal is input will stop.
- (2) In the automatic mode, all axes will stop when the interlock signal is input to even one axis which coincides with the moving axis.
- (3) 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.

(Note 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.

- (4) 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.

(Note 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

M system	○
L system	○

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.

15.3.9 Door Interlock

15.3.9.1 Door Interlock I

M system	○
L system	○

(Outline of function)

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:

- (1) It stops operations.
 - (a) 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.
 - (b) When manual operation was underway
All the axes decelerate and stop immediately.
The spindle also stops.
- (2) The complete standby status is established.
- (3) After all the NC axes and the spindle have stopped, the ready OFF status is established.
- (4) 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.

- (5) All the axes are set to ready ON.
- (6) The door open enable signal is set to OFF.

Resuming operation

- (7) When automatic operation was underway
Press the AUTO START button.
Operation now resumes from the block in which machining was suspended when the door open signal was input.
- (8) When manual operation was underway
Axis movement is commenced when the axis movement signals are input again.
- (9) 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.

15.3.9.2 Door Interlock II

M system	○
L system	○

Outline of function

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, automatic 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:

- (1) It stops operations.
All the axes decelerate and stop.
The spindle also stops.
- (2) The complete standby status is established.
- (3) After all the NC axes and the spindle have stopped, the ready OFF status is established.
However, the servo ready finish 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.

- (4) All the axes are set to ready ON.
- (5) The door open enable signal is set to OFF.

Resuming operation

- (6) 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.
- (7) When manual operation was underway
Axis movement is commenced when the axis movement signals are input again.
- (8) 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) 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.

Differences from door interlock I

- (1) The method used to stop the machine during automatic operation is the same as with the axis interlock function.
- (2) The servo ready finish signal (SE) is not set to OFF.
- (3) Automatic start is valid during door interlock. However, the interlock takes effect for the axis movements.
- (4) When this door interlock function (door open signal ON) is initiated during axis movement, the axes decelerate and stop.
- (5) When this door interlock function (door open signal) is set to OFF, the axis movement resumes.

15.3.10 Parameter Lock

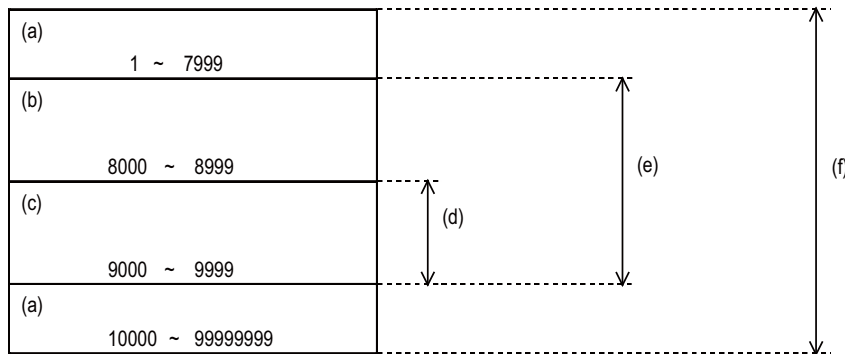
M system	○
L system	○

This function is used to prohibit changing the machine parameter.

15.3.11 Program Protection (Edit Lock B, C)

M system	○
L system	○

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.



- (a) Machining program A
- (b) Machining program B (User-prepared standard subprogram)
- (c) Machining program C (Machine tool builder customized program)
- (d) Editing is inhibited by edit lock C.
- (e) Editing is inhibited by edit lock B.
- (f) Editing is inhibited by data protect.

15.3.12 Program Display Lock

M system	○
L system	○

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.

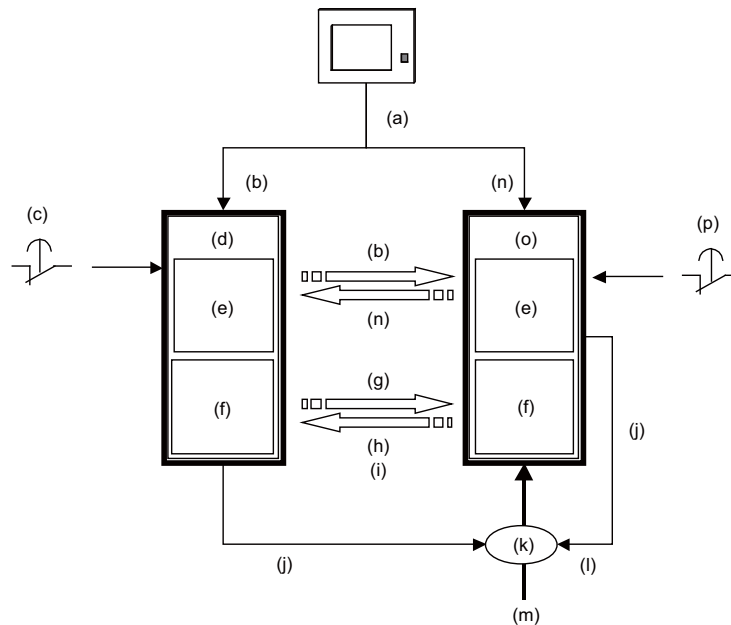
- 0: Display and search are possible.
- 1: Display of the program details is prohibited.
- 2: Display and operation search of the program details are prohibited.

15.3.13 Safety Observation

M system	○
L system	○

With the safety observation function, access to the working part of the machine without turning the motor drive power OFF is possible even when the machine door is open. Operation speed will be the pre-set safety speed or lower while the door is open.

Function	Description
Dual emergency stop	The control unit and the drive unit separately observe the input of emergency stop. In case of an emergency stop, both the control unit and the drive unit (power supply unit) will independently control the contactors to shut the drive power OFF.
Speed monitor	The 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. (1) That the command speed should not exceed the speed set by the parameter (the "safety speed"). (2) That the motor rotation speed should not exceed the rotation speed set by the parameter (the "safety rotation speed").
Door state signal monitor	The control unit and the drive unit separately observe the door state signal (open or closed).



- | | |
|----------------------------------|----------------------------------|
| (a) Door open/close check sensor | (b) NC-side door state signal |
| (c) External emergency stop A | (d) NC |
| (e) Door state signal monitor | (f) Speed monitor |
| (g) Commanded position | (h) Feedbacked position |
| (i) Feedbacked rotation speed | (j) Contactor control |
| (k) MC | (l) Contactor |
| (m) Main power | (n) Drive-side door state signal |
| (o) Drive | (p) External emergency stop B |

15.3.14 Vertical Axis Pull-up

M system	○
L system	○

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 μ m 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.

15.4 Maintenance and Troubleshooting

15.4.1 Operation History

M system	○
L system	○

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 two types of history data files are used.

- All history: The key history, alarm/warning history, PLC signal history, and AC input power OFF history is recorded in date/time order.
- Key history: Only the key history is recorded in date/time order.

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) If the date and time are the same, the history will be displayed in the following order.

- Key history
- Alarm/warning history
- PLC signal history
- AC input power OFF history

15.4.2 Data Sampling

M system	<input type="radio"/>
L system	<input type="radio"/>

The NC data sampling function samples the NC internal data (speed output from NC to drive unit, and feedback data from the drive unit, etc.). This data can be output as text data.

Item	Specifications
Sampling cycle	1.7ms × setting value
Number of sampled axes	NC axis : 1 to 3 axes Spindle : 1 to 2 axes
Number of sampled channels	1 to 8 points
Sampling data size	Maximum 1,310,720 points (Note 1) This is the entire data size. The data size per channel will decrease when the number of sampled channels increases. (Note 2) If the open DRAM memory is insufficient, the maximum data size will decrease.

- The data set with this function is not output to the parameters.
- The state returns to "Sampling stop" when the power is turned ON.

15.4.3 NC Data Backup

M system	<input type="radio"/>
L system	<input type="radio"/>

The NC data back up function backs up the NC parameters, etc., on an IC card.
The data can also be restored.

15.4.5 Servo Tuning Support Tool

15.4.5.2 NC Analyzer

M system	<input type="radio"/>
L system	<input type="radio"/>

(Note) Please contact us to purchase this tool.

Servo parameters can be automatically adjusted by activating the motor using machining programs for adjustment or vibration signals, and measuring/analyzing the machine characteristics.

(To be supported by NC Analyzer version B1.)

15.4.6 Automatic Backup

M system	<input type="radio"/>
L system	<input type="radio"/>

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.

15.4.7 System Setup

M system	<input type="radio"/>
L system	<input type="radio"/>

System setup function enables automatic settings for the NC's initial startup just by inputting the minimally required items.

With this function, settings required for preliminary operation of the motors (servo motor, spindle motor and auxiliary axis motor) can be provided easily, resulting in shortening of the time required for the NC initial startup.

The required setting items for this function are as follows.

- (1) Number of spindle connections, number of auxiliary axis connections
- (2) Command type
- (3) Setting of the servo interface connection channel and rotary switch for each axis drive. Also, converter /motor type connected with each spindle drive.
- (4) Setting of the servo interface connection channel and rotary switch for each NC axis. Also, the motor type, encoder type, and the converter type that is connected with the servo drive.

By setting above items, the following setting required for the NC's initial startup can be executed.

- (1) SRAM clear and file format
- (2) Various parameter settings (machine parameter and user parameter)
 Note that there are two methods of setting spindle parameters: using spindle parameter files provided by Mitsubishi, and setting in system setup screen.
- (3) PLC ladder writing settings required for the manual operation (JOG mode only)
 Note that a menu has to be selected again.

15.4.13 Parameter Setting Tool

15.4.13.1 NC Configurator2

M system	<input type="radio"/>
L system	<input type="radio"/>

The NC data file necessary for NC control and machine operation (such as parameters, tool data and common variables) can be edited on a personal computer.

Please contact us to purchase a full function version. A limited function version is also available free of charge.

Drive System

16.1 Servo/Spindle

Refer to "CONTENTS II Functional Specifications MITSUBISHI CNC E70 Series Specifications List" for details.

Refer to the following manual for details of Servo and Spindle system.

Drive System Data Book (MDS-D2/DH2/DM2/DJ Series)(IB-1501142)

MDS-DJ Series Specifications Manual (IB-1501130)

MDS-DM2 Series Specifications Manual (IB-1501136)

Machine Support Functions

17.1 PLC

17.1.1 Built-in PLC Processing Mode

M system	○
L system	○

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.

(1) High-speed processing

This mode provides repeated execution at 3.5ms cycles. It is used to process signals requiring high speeds. The max. number of program steps for high-speed processing (1 period) is 1000 steps when using basic commands.

(2) 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

M system	○
L system	○

(1) Ladder commands

Basic commands (bit processing commands)

43 commands including LD, LDI, OR, ORI, AND, ANI, OUT, PLS, etc.

Function commands

188 commands including data transfer, 4 basic arithmetic operations, logic arithmetic operations, large/small identification, binary/BCD conversion, branching, conditional branching, decoding, encoding, etc.

17.1.3 PLC Support Functions

17.1.3.1 Alarm Message Display

M system	○
L system	○

The contents of the alarms which have occurred during sequence (user PLC) processing 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

M system	<input type="radio"/>
L system	<input type="radio"/>

When a condition has arisen in which a message is to be relayed to the operator, an operator message can be displayed separately from the alarm message.

The maximum length of an operator message on the alarm diagnosis screen is 60 characters. The number of messages displayed at the same time is one.

17.1.3.3 Memory Switch (PLC Switch)

17.1.3.3.1 PLC Switch 32 Points

M system	<input type="radio"/>
L system	<input type="radio"/>

32 points of PLC switches can be set on the setting and display unit screen, and the ON/OFF control executed.

The switches can be used as part of the machine operation switches. The switch applications can be freely determined with the sequence program, and each switch name can be created with the PLC and displayed on the setting and display unit.

17.1.3.4 Load Meter Display

M system	<input type="radio"/>
L system	<input type="radio"/>

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

M system	<input type="radio"/>
L system	<input type="radio"/>

The user PLC version can be displayed in the software list on the Software Configuration screen.

17.1.3.6 Multi-ladder Program Register and Execution

M system	<input type="radio"/>
L system	<input type="radio"/>

Up to 20 sequence programs can be registered. Only one execution type can be set in one program. The following five execution types are available

- "Initial" (Initialization process) : Started only once when power is turned ON.
- "Scan" (High-speed process) : Started at the standard interrupt cycle.
- "Scan" (Main process) : Constantly started in process other than high-speed process.
- "Standby" (Standby process) : Called from the high-speed process or main process.
- "Low-speed" : This execution type is not used.

17.1.3.7 Ladder Program Writing during RUN

M system	<input type="radio"/>
L system	<input type="radio"/>

Ladder program can be edited while PLC is running. This function is available, either by GX Developer or PLC onboard edit.

17.1.3.8 PLC Protection

M system	<input type="radio"/>
L system	<input type="radio"/>

A keyword can be set to protect the sequence programs stored in CNC.

The following two types of conditions can be selected at the keyword registration.

- Write protection: Writing and editing are restricted.
- Read/Write/Display protection: Writing, reading, verification and list display are restricted.

For details, refer to the PLC Programming Manual.

17.1.4 Built-in PLC Capacity [Steps]


17.1.4.1 Standard PLC Capacity

M system	<input type="radio"/> 8000
L system	<input type="radio"/> 8000

17.1.5 Machine Contact Input/Output I/F

M system	<input type="radio"/>
L system	<input type="radio"/>

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 "Hardware Specifications" in the "General Specifications" for details.

Up to 2 channels and 3 handles can be used for the remote type machine contact input.

17.1.6 Ladder Monitor

M system	<input type="radio"/>
L system	<input type="radio"/>

This function enables the operating status of the sequence circuit to be checked on the controller's setting and display unit.

The monitor functions include the following.

- (1) Circuit monitoring
- (2) Screen stop by monitor stop trigger point
- (3) Entry monitoring
- (4) Decimal-hexadecimal conversion present value monitoring

17.1.7 PLC Development

17.1.7.1 On-board Development

M system	<input type="radio"/>
L system	<input type="radio"/>

On-board refers generically to the PLC related operations carried out with the CNC unit.

The Mitsubishi CNC M70V Series on-board realizes functions and operations similar to the MELSEC Series ladder development tool (GX Developer).

This improves the user's ladder editing operations, and provides an easy-to-use onboard with powerful ladder monitoring functions.

17.1.7.2 MELSEC Development Tool (GX Developer)

M system	<input type="radio"/>
L system	<input type="radio"/>

This function enables the data of the PLC contained inside the NC system to be developed and debugged using the GX Developer installed in a personal computer (OS:Windows).

Many and varied functions of the GX Developer make it possible to reduce the PLC data development and debugging time.

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)

M system	<input type="radio"/>
L system	<input type="radio"/>

17.2 Machine Construction

17.2.1 Servo OFF

M system	○
L system	○

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.)

- (1) The counter is corrected according to the movement amount (follow up function).
- (2) 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

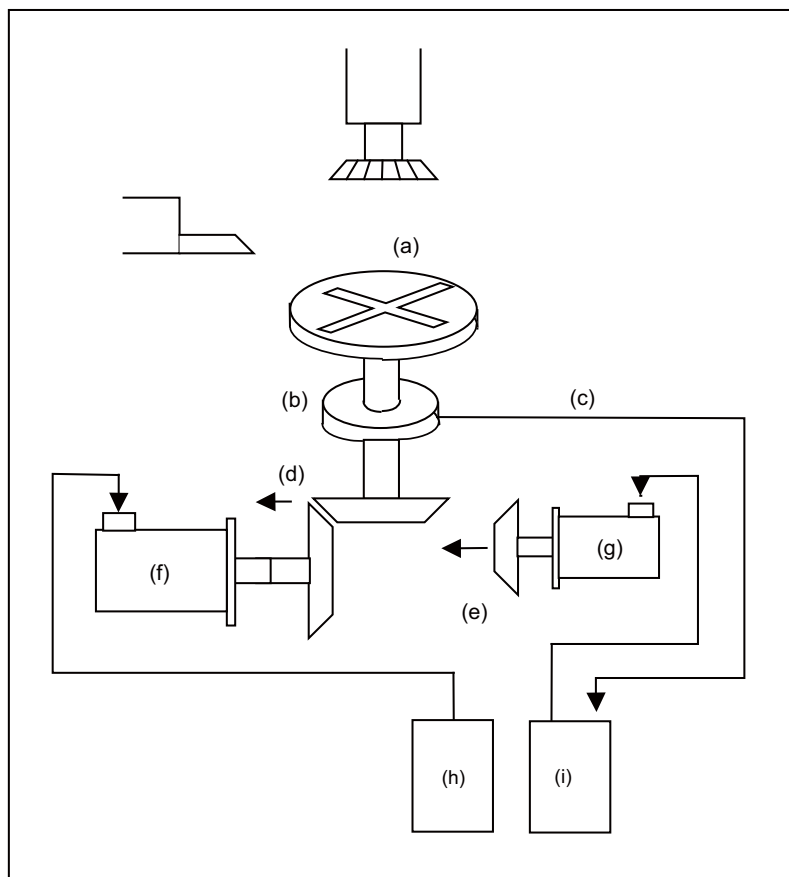
Even if the servo OFF axis is moved with the mechanical handle with the application of the servo OFF function and follow up function, the position data can be constantly read in and the machine position updated. Thus, even if the axis is moved with the mechanical handle, the coordinate position display will not deviate.

17.2.2 Axis Detachment

M system	<input type="radio"/>
L system	<input type="radio"/>

This function enables the control axis to be freed from control. Conversely, an axis which has been freed from control can be returned to the control status.

This function enables the rotary table or attachments to be removed and replaced. Automatic operation is disabled until the axis for which the axis detach command has been released completes its dog-type reference position return.



This shows the configuration of a machine for which switching between the C axis and turning table is performed. When the spindle motor is connected, the C axis is placed in the detached status.

As a result, the position feedback of the detector is ignored.

- (a) C-axis/turning table
- (b) Rotary magnetic scale
- (c) (Position feedback)
- (d) (OFF with C-axis control)
- (e) (Coupled with C-axis control)
- (f) Spindle motor
- (g) C-axis motor
- (h) Spindle drive unit
- (i) C-axis drive unit

POSITION X 123.456 Z 0.000#1 C 345.678><	The detached status > < is indicated on the right of the POSITION display on the POSITION screen and at the same time the servo ready for the controller output signal is set to OFF. The POSITION counter retains the value applying when detach was assigned.
---	---

(Note) Axis detach can be executed even for the absolute position detection specifications axis, but when the axis is reinstalled, the zero point must be set.

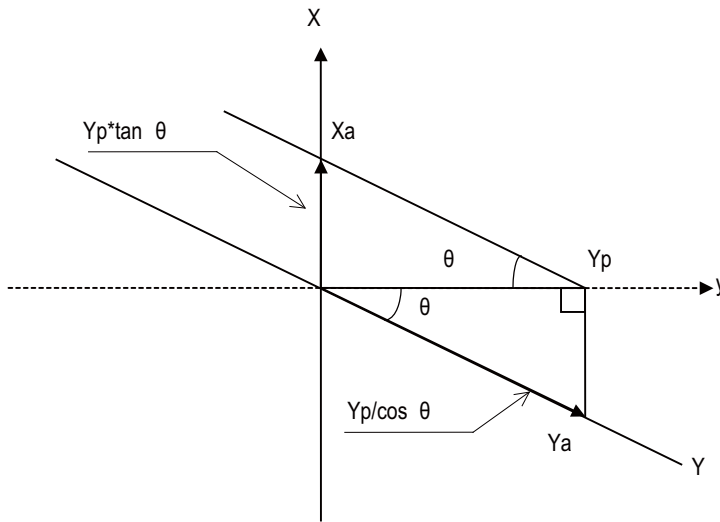
17.2.4 Inclined Axis Control

M system	—
L system	○

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.

<Example of use> When 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:

$$Ya = Yp / \cos \theta \dots\dots (1)$$

The X-axis (basic axis) movement amount is compensated by the inclined movement of the Y axis, and it is expressed as follows:

$$Xa = Xp - Yp * \tan \theta \dots (2)$$

The Y-axis (inclined axis) speed is as follows:

$$Fa = Fp / \cos \theta$$

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

M system	○ 24
L system	○ 24

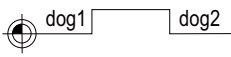

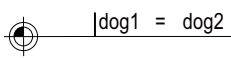
The position switch (PSW) function provides hypothetical dog switches in place of the dog switches provided on the machine axes by setting the axis names and coordinate positions indicating the hypothetical dog positions as parameters beforehand so that signals are output to the PLC interface when the machine has reached these hypothetical dog range. The hypothetical dog switches are known as position switches (PSW).

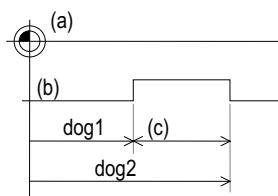
The coordinate position indicating the hypothetical dog positions (dog1, dog2) on the coordinate axes whose names were set by parameters ahead of time in place of the dog switches provided on the machine axes are set using position switches. When the machine has reached the hypothetical dog range, a signal is output to the device supported by the PLC interface.

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

dog1, dog2 settings	dog1, dog2 positions	Description
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



- (a) Basic machine coordinate system zero point
- (b) Hypothetical dog
- (c) Position switch area (PSW width)

17.3 PLC Operation

17.3.1 Arbitrary Feed in Manual Mode

M system	○
L system	○

This function enables the feed directions and feed rates of the control axes to be controlled using commands from the user PLC.

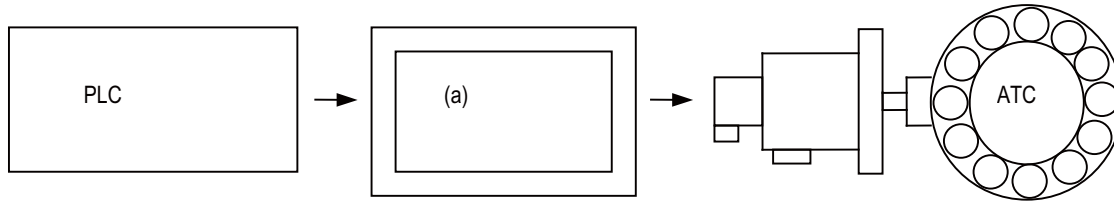
The arbitrary feed function controls the movement of the axes at the specified rates while the start signal is output from the PLC to the NC system.

PLC operations can be performed even during manual operation or automatic operation, but they cannot be performed when an axis for which arbitrary feed has been assigned is executing a command from the NC system (that is, while the axis is moving).

17.3.3 PLC Axis Control

M system	○
L system	○

Over and above the NC control axes, this function enables axes to be controlled independently by commands based on the PLC.



(a) PLC axis control

Item	Description							
Number of control axes	Max. 2 axes							
Simultaneously controlled axes	PLC control axis is controlled independently from NC control axes. A multiple number of PLC axes can be started simultaneously.							
Command increment	Least command increment 0.001 mm (0.0001 inch) 0.0001 mm (0.00001 inch) (Same as command increment for NC control axes)							
Feed rate	0 to 1000000mm/min (0 ~ 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 (0.001mm/0.0001inch)							
Operation modes	Rapid traverse, cutting feed, jog feed (+) (-), reference position return feed (+) (-), handle feed							
Acceleration/deceleration	<table style="border: none; width: 100%;"> <tr> <td style="border: none;">Rapid traverse, jog feed</td> <td rowspan="2" style="border: none; vertical-align: middle;">} Linear acceleration/deceleration</td> </tr> <tr> <td style="border: none;">Reference position return feed</td> </tr> <tr> <td style="border: none;">Cutting feed</td> <td style="border: none;">} Exponential function acceleration/deceleration</td> </tr> <tr> <td style="border: none;">Handle feed</td> <td style="border: none;">} Step</td> </tr> </table>	Rapid traverse, jog feed	} Linear acceleration/deceleration	Reference position return feed	Cutting feed	} Exponential function acceleration/deceleration	Handle feed	} Step
Rapid traverse, jog feed	} Linear acceleration/deceleration							
Reference position return feed								
Cutting feed	} Exponential function acceleration/deceleration							
Handle feed	} Step							
Backlash compensation	Available							
Stroke end	None							
Soft limit	Available							
Rotary axis command	Available For absolute commands.....amount within 1 rotation (rotation by amount remaining after division into 360) For incremental commands.....rotation 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

M system	○
L system	○

This function is used to move the PLC axis to the position (station) registered on the table.
The destination (station) can be determined by equally dividing the valid axis stroke or by using an arbitrary coordinate that has been stored in advance. The PLC program can also be used to specify arbitrary values of the coordinate.

(1) 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.

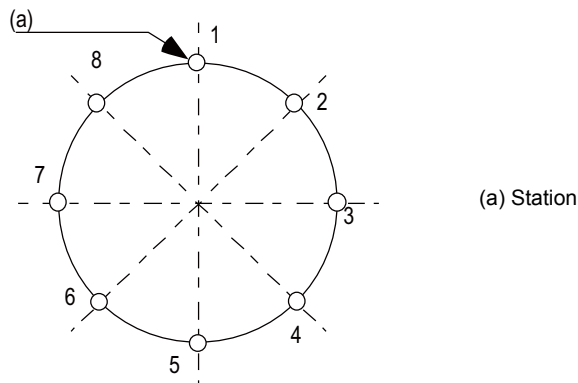


Figure 1. Setting 8 stations (8 divisions)

Station method (for linear axis) : A valid stroke is equally divided to determine the stations (positioning destinations). The maximum number of divisions is 359.

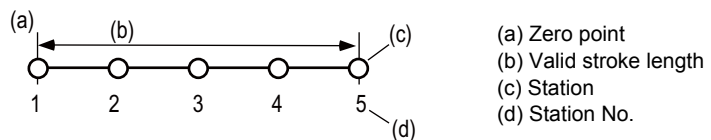


Figure 2. Setting 5 stations

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".

(2) Feed functions

Feed rate selection : Four different feed rates are set to be selected with the PLC program.

Acceleration/deceleration pattern : Four different acceleration/deceleration patterns are set to be selected with the PLC program. The patterns are all constant inclination acceleration/deceleration, which have options of linear or soft acceleration/deceleration.

Short-cut control : A least movement distance is automatically judged when a rotary axis is rotated.

(3) Operation functions

The following operation modes are available. Send a command from PLC to change the operation mode.

- Automatic 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.
- Manual 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.
- Manual handle mode : The axis is moved by the manual PLG.

(4) Operation support function

- Position switch : A signal is output to the PLC interface when the machine has reached within the specified range.

17.4 PLC Interface

17.4.1 CNC Control Signal

M system	O
L system	O

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.

(1) 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

(2) 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)

(3) 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

M system	○
L system	○

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 operation ready

When the controller power is turned ON and the servo system enters the operation ready status, the "Servo ready" 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" 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 automatic 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 automatic 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 automatic 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.

- (a) G84, G88 (fixed cycle: tapping cycle)
- (b) G84.1, G88.1 (fixed cycle: reverse tapping cycle)
- (c) 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.

(a) Automatic mode

The signal is output in the movement command of each axis. It is output until the machine stops during stop based on feed hold or block stop.

(b) 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.

(c) 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.

(a) System errors**(b) Servo alarms****(c) Program errors****(d) 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 finish

In the memory or MDI automatic operation, the "Movement command finish" 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

M system	<input type="radio"/>
L system	<input type="radio"/>

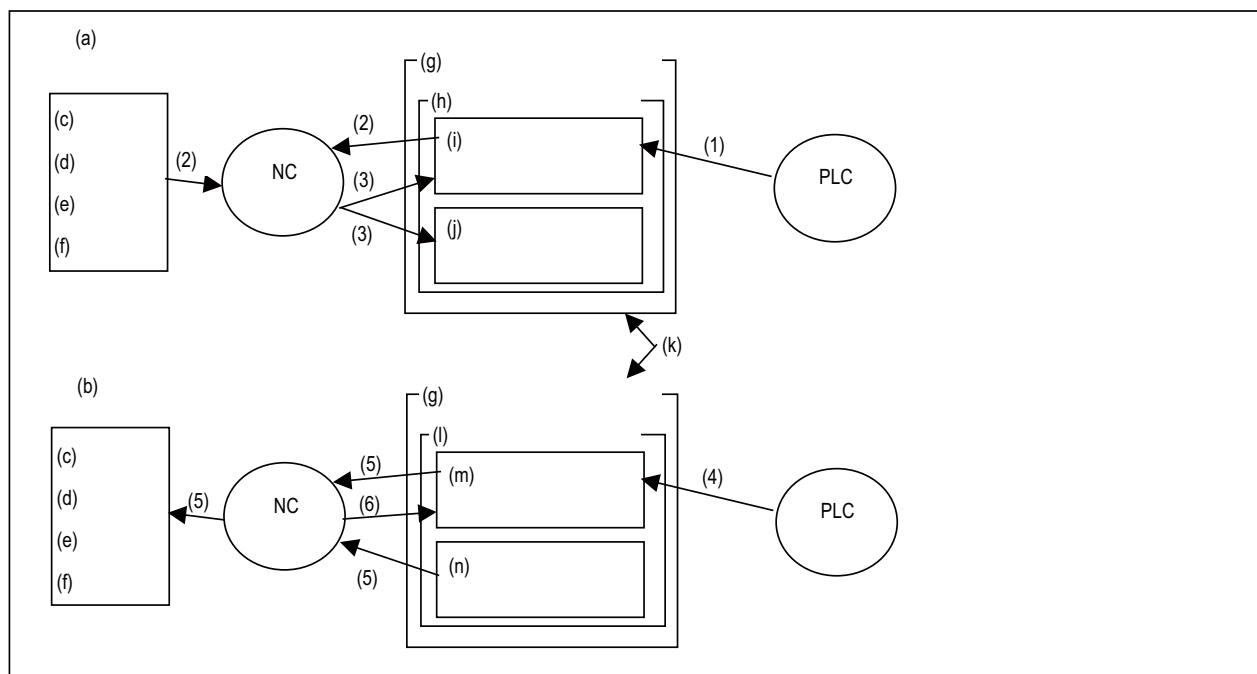
This function uses the "read window" or "write window" assigned to the R register's user area to read and write the CNC operation status, axis information, parameters and tool data, etc.

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 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.



- (a) <Reading>
- (b) <Writing>
- (c) Operation status
- (d) Axis information
- (e) Parameters
- (f) Tool data
- (g) R register (user area)
- (h) Read window
- (i) Read control window
- (j) Read data window
- (k) Designate arbitrary position with parameter (R register)
- (l) Write window
- (m) Write control window
- (n) Write data 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

M system	<input type="radio"/>
L system	<input type="radio"/>

The program which is automatically operated can be searched from PLC. The program No., sequence No. and block No. can be assigned. Also the content currently searched can be read.

17.5 Machine Contact I/O

17.5.1 Additional DI/DO (DI: 32/DO: 32)

M system	<input type="checkbox"/>
L system	<input type="checkbox"/>

17.5.2 Additional DI/DO (DI: 64/DO: 64)

M system	<input type="checkbox"/>
L system	<input type="checkbox"/>

17.5.3 Remote IO 32/32

M system	<input type="checkbox"/>
L system	<input type="checkbox"/>

17.5.4 Remote IO 64/48

M system	<input type="checkbox"/>
L system	<input type="checkbox"/>

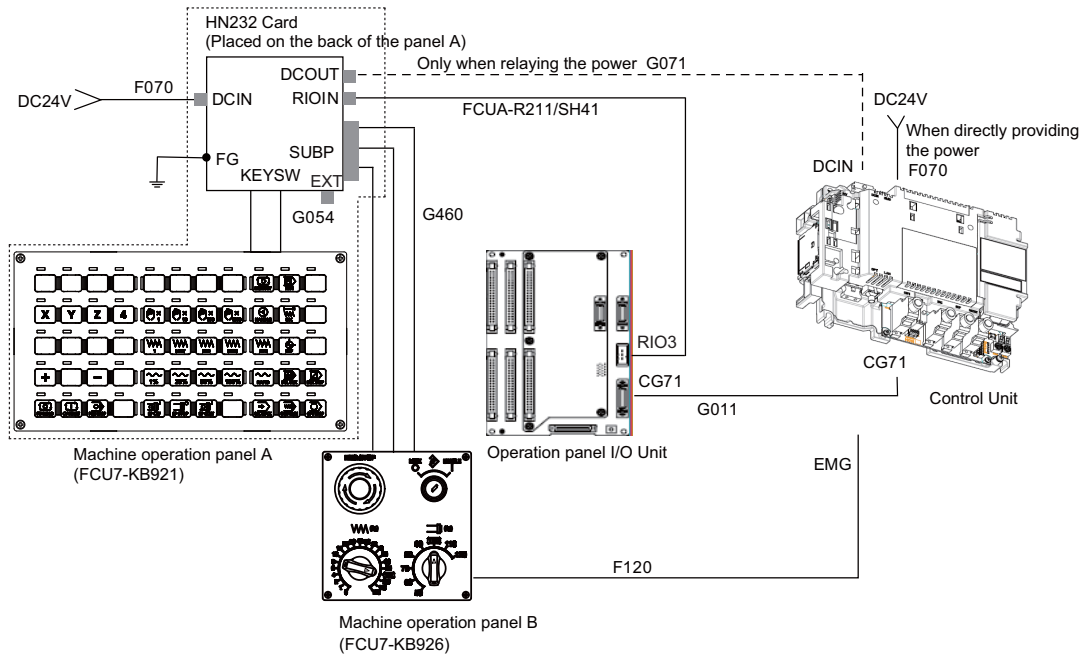
17.5.5 MITSUBISHI CNC Machine Operation Panel

M system	△
L system	△

MITSUBISHI CNC machine operation panel can change its keyboard layout according to the specifications of the machine.

Also, a sample of the PLC program supporting the basic key board layout is available. By adding an interface to the machine, a PLC program which fits the machine can easily be created.

Wirings have been reduced as a result of directly connecting with the NC via the remote I/O link communication.



17.6 External PLC Link

17.6.3 CC-Link (Master/Local)

M system	□ (*7)
L system	□ (*7)

(*7) When HN746 is installed.

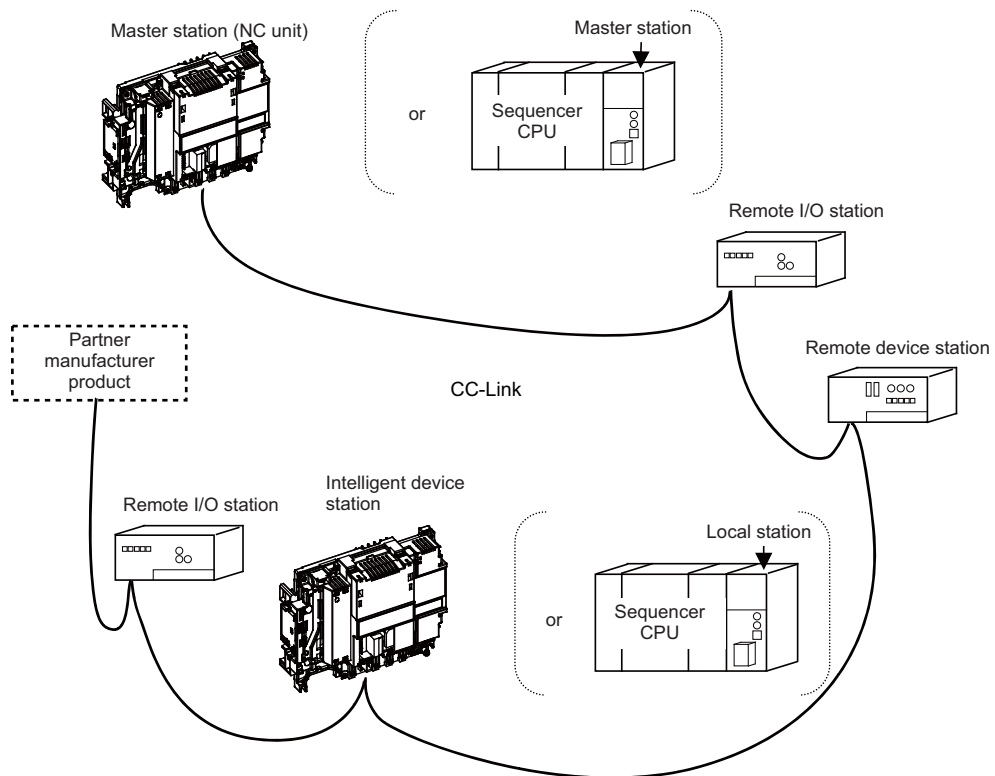
NC unit can be directly connected to the network to serve as the master/local station of the MELSEC CC-Link. CC-Link unit is required for this connection.

With this function, the GOT connection and the transient instruction with MELSEC A series cannot be used.

(1) 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 sequencer CPUs or NCs.
- Connecting the device equipments made by the partner manufacturer can flexibly support various systems.

(2) Outline drawing



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)

- (3) Performance specifications
The E70 Series is compliant with CC-Link Ver.2.00.
<CC-Link Ver.2.00 specification>

Item		CC-Link master/local unit				
Max. number of link points per one system (Note 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 one 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	156kbps / 625kbps / 2.5Mbps / 5Mbps / 10Mbps 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 method	Bus (EIA RS485 compliant)					
Transmission format	HDLC compliant					
Illegal control method	$CRC(X^{16} + X^{12} + X^5 + 1)$					

Item	CC-Link master/local unit																		
Max. number of connection units	64 units Note that the following two conditions must be satisfied. Condition 1: - $(a+a2+a4+a8)+(b+b2+b4+b8) \times 2+(c+c2+c4+c8) \times 3+(d+d2+d4+d8) \times 4 \leq 64$ - $(a \times 32+a2 \times 32+a4 \times 64+a8 \times 128)+(b \times 64+b2 \times 96+b4 \times 192+b8 \times 384)+(c \times 96+c2 \times 160+c4 \times 320+c8 \times 640)+(d \times 128+d2 \times 224+d4 \times 448+d8 \times 896) \leq 8192$ - $(a \times 4+a2 \times 8+a4 \times 16+a8 \times 32)+(b \times 8+b2 \times 16+b4 \times 32+b8 \times 64)+(c \times 12+c2 \times 24+c4 \times 48+c8 \times 96)+(d \times 16+d2 \times 32+d4 \times 64+d8 \times 128) \leq 2048$ 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: - $\{(16 \times A)+(54 \times B)+(88 \times C)\} \leq 2304$ - 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																		
Remote station No.	1 to 64																		
Max. total cable length and each cable length between stations	CC-Link Ver.1.10 compliant cable (When a 110Ω of terminating resistance is used.) (Note 2) <table border="1" style="width: 100%; border-collapse: collapse;"> <thead> <tr> <th style="width: 30%;">Baud rates</th> <th style="width: 40%;">Cable length between stations</th> <th style="width: 30%;">Max. total cable length</th> </tr> </thead> <tbody> <tr> <td>156kbps</td> <td></td> <td>1200m</td> </tr> <tr> <td>625kbps</td> <td></td> <td>900m</td> </tr> <tr> <td>2.5Mbps</td> <td>20cm or more</td> <td>400m</td> </tr> <tr> <td>5Mbps</td> <td></td> <td>160m</td> </tr> <tr> <td>10Mbps</td> <td></td> <td>100m</td> </tr> </tbody> </table>	Baud rates	Cable length between stations	Max. total cable length	156kbps		1200m	625kbps		900m	2.5Mbps	20cm or more	400m	5Mbps		160m	10Mbps		100m
Baud rates	Cable length between stations	Max. total cable length																	
156kbps		1200m																	
625kbps		900m																	
2.5Mbps	20cm or more	400m																	
5Mbps		160m																	
10Mbps		100m																	
Connection cable	CC-Link Ver.1.10 compliant cable (3-core twisted pair cable with shield) (Note 3)																		
RAS function (Note 4)	- Automatic link refresh function - Sub-station isolation function - Link special relay/error detection by register																		

- (Note 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 max. number of link points per one system. This applies for the remote register (RWw, RWr), as well.
- (Note 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.
- (Note 3) This can be used with the cables made by different manufacturers as long as they are Ver.1.10 compliant cables.
- (Note 4) "RAS" is short for "Reliability", "Availability" and "Serviceability".

(4) Usable functions

Of all the CC-Link functions, the following functions can be used with the NC.

(○ : Available ×: Not available – : Not relevant)

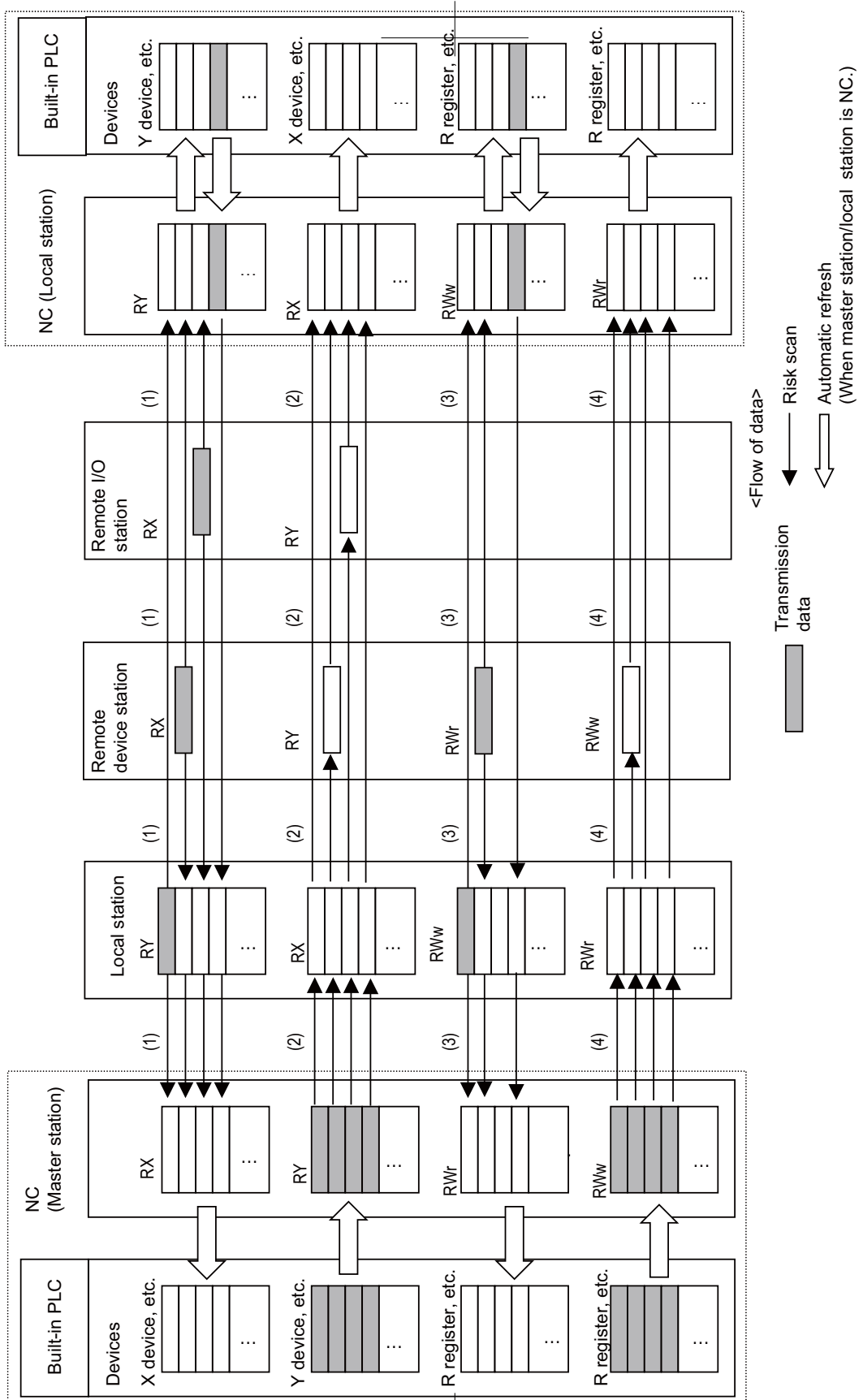
	Function item	MELSEC		NC unit	
		Master station	Local station	Master station	Local station
Method	Ver.1.00	○	○	○	○
	Ver.2.00	○	○	○	○
Master function	Communication between master station and remote I/O station	○	–	○	–
	Communication between master station and remote device station	○	–	○	–
	Communication 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 E ² PROM	○	–	○ (Note 2)	–
	Setting of input data status from data link trouble station	○	○	○	○
	Unit resetting by sequence program	○	○	○	○
	Data link stop/restart	○	○	○	○
	Parameter registration function	○	○	○ (Note 2)	○ (Note 2)
	Automatic refresh function	○	○	○	○
	Scan synchronization function	Synchronous mode	○	–	○
Asynchronous mode		○	○	○	○
Setting & display function	LED diagnosis status	16-point display (A1SJ61QBT11)		16-point display	
	Station number setting	Unit front panel switches		Parameters (Note 2)	
	Baud rate setting				
	Mode setting switch				
	Condition setting			H/W switches	
RAS function	Automatic link refresh function	○	○	○	○
	Sub-station isolation function	○	–	○	–
	Data link status check (SB/SW)	○	○	○	○
	Off-line test	○	○	○	○
	On-line test	○	○	○	○
	Monitor diagnosis	○	○	×	×
	Standby master function (Note 3)	○	–	○	–
	Temporary error cancel station designation function	○	–	○	–
Dedicated instruction	READ instruction / SREAD instruction (Note 1)	○	○	○	○
	WRITE instruction / SWRITE instruction (Note 1)	○	○	○	○
	RIRD instruction / RIWT instruction (Note 1)	○	○	×	×

(Note 1) The transient instruction cannot be used.

(Note 2) The parameter for the CC-Link is set with the GX Developer.

(Note 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. (The NC unit cannot be used as the standby master station.)

(5) 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.)



- (1) 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).
- (2) 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).
- (3) By executing a link scan, data in the remote device station's remote register (RW_r) and the local station's remote register (RW_w) is transmitted to the master station's remote register (RW_r) and the local station's remote register (RW_w).
- (4) By executing a link scan, data in the master station's remote register (RW_w) is transmitted to the remote device station's remote register (RW_w) and the local station's remote register (RW_r).

17.7 Installing S/W for Machine Tools

17.7.1 Customization (NC Designer)

M system	○
L system	○

Custom release is an optional function and allows the user-original window to display as an HMI screen or another application.

(Note) Please contact us to purchase this tool.

Custom release includes, mainly, F0 release and menu release:

F0 release : Custom release screen (Note 1) can be assigned to function keys (F0, SEP, window display, window selection).

When a function key is pressed, the assigned custom release screen will be displayed.

"NC Designer interpreter method", "NCDesigner compilation method" and "Executing file registration method" can be used for registration.

Menu release : Custom release window (Note 2) can be registered in the main menu of the monitor screen, setup screen and edit screen.

Main menu contents of the monitor, setup and edit screen can be rearranged.

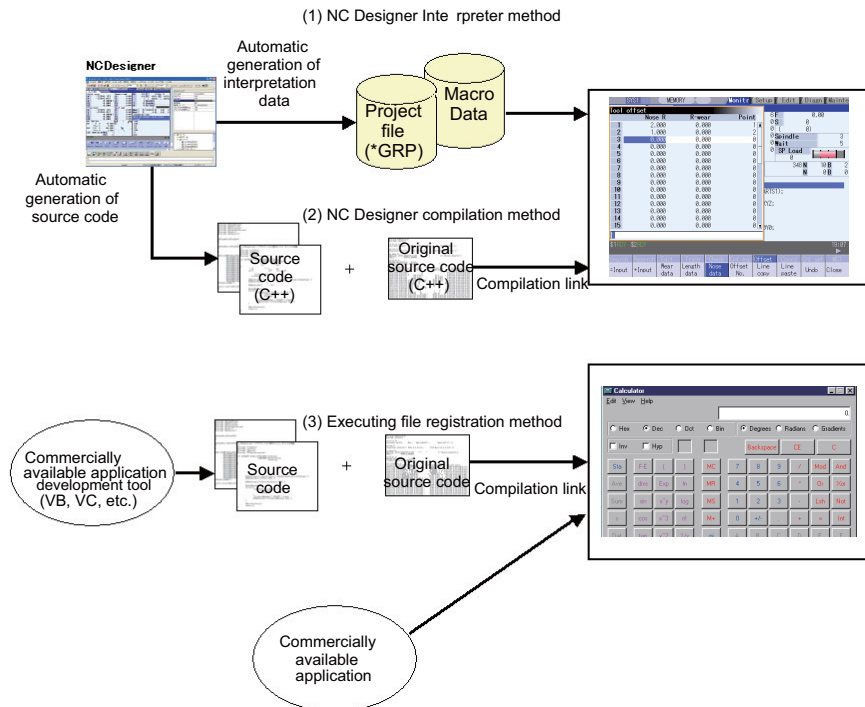
When the main menu in which custom release window is registered is pressed, the custom release window will be displayed.

"NC Designer interpreter method", "NC Designer compilation method" and "Executing file registration method" can be used for registration.

Depending on the conditions, display/non-display of the custom menu can be changed.

(Note 1) An HMI screen originally created with NC Designer by the user or an execution file prepared by the user.

(Note 2) An HMI window originally created with NC Designer by the user or an execution file prepared by the user.



17.7.1.1 Customization Data Storage Capacity [MByte]

M system	1
L system	1

17.7.1.2 Customization Working Memory Size [MByte]

M system	1
L system	1

17.7.1.3 Direct Screen Selection

M system	<input type="radio"/>
L system	<input type="radio"/>

17.7.3 EZSocket I/F

M system	<input type="radio"/>
L system	<input type="radio"/>

(Note) Please contact us to purchase this tool.

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

M system	<input type="radio"/>
L system	<input type="radio"/>

(Note) Please contact us to purchase this tool.

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. The processing frequency is 7.1 (ms).

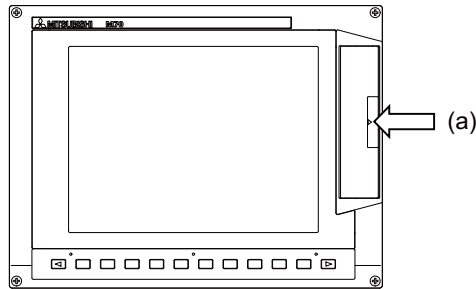
- Type 2 : By registering the start address of the module to be processed synchronously with PLC, C language module will be started synchronizing with PLC. Note that C language will not be called if PLC is incorrect, even if the process has been registered.

- 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.

[Hardware configuration]

This function will be activated by installing C language module into a compact flash or built-in F-ROM. Compact flash interface is located on the control unit. Up to 120KB data can be stored in built-in F-ROM. When C language module is installed in compact flash, C language inside the compact flash is executed. (The module inside the built-in F-ROM is not executed.)

There is a list of compact flash cards which are tested by us in "5 Hardware Specifications" of "Precautions for Use of Commercially Available CF Cards".



(a) Compact flash interface

[Software configuration]

The names of directory, file and initialize function, where C language modules are stored, are fixed.

(Note) Incorrect hardware or software configuration disables operations regardless of option function.

17.8 Others

17.8.1 System Lock

M system	<input type="radio"/>
L system	<input type="radio"/>

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) We do not compensate for any detriment that may be caused when the lock is cancelled illegally.

Revision History

Date of revision	Manual No.	Revision details
Nov. 2012	IB(NA)1501144-A	First edition created.
Dec. 2012	IB(NA)1501144-B	Mistakes were corrected.
Dec.2013	IB(NA)1501144-C	<p>Revisions were made to support E70 S/W version K0. The following chapters were added. " II Functional Specifications" -8.3.6 Miscellaneous function command high-speed output The following chapters were revised. - 4.4.1 Automatic acceleration/deceleration after interpolation - 9.3.2.1 Tool shape/wear offset amount - 12.1.1.1 Subprogram control - 12.1.2.1 User macro - 12.1.2.3 Macro interruption - 12.1.3.5 Compound type fixed cycle for turning machining - 12.1.6.1 Corner chamfering/Corner R - 12.1.9.2 Compensation data input by program - 14.1.3 Manual tool length measurement 1 - 14.2.1.1 Tool life management I - 15.3.2.1 Stored stroke limit I/II - 15.4.1 Operation history - 16.1 Operation history - 17.2.5 Position switch Other contents were added/revised/deleted according to specification.</p>
Sep.2015	IB(NA)1501144-D	<p>Revisions were made to support E70 S/W version L0. The following chapters were added. " II Functional Specifications" - 12.4.3 Simple Programming - 17.6 External PLC Link - 17.6.3 CC-Link (Master/Local) - 17.7.1.3 Direct Screen Selection</p> <p>The following chapters were revised. "I General Specifications" - 2 General Connection Diagram - 3.1 List of Units - 4 Environment Conditions - 5.1 Control Unit</p> <p>" II Functional Specifications" - 1.1.2.3 Max. Number of PLC Axes - 6.2.11 Parameter/Operation Guidance - 6.2.12 Alarm Guidance - 12.4.4 G code Guidance</p> <p>Mistakes were corrected.</p>

Global Service Network

AMERICA

MITSUBISHI ELECTRIC AUTOMATION INC. (AMERICA FA CENTER)

Central Region Service Center
500 CORPORATE WOODS PARKWAY, VERNON HILLS, ILLINOIS 60061, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-847-478-2650

Michigan Service Satellite
ALLEGAN, MICHIGAN 49010, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-847-478-2650

Ohio Service Satellite
LIMA, OHIO 45801, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-847-478-2650
CINCINNATI, OHIO 45201, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-847-478-2650

Minnesota Service Satellite
ROGERS, MINNESOTA 55374, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-847-478-2650

West Region Service Center
16900 VALLEY VIEW AVE., LAMIRADA, CALIFORNIA 90638, U.S.A.
TEL: +1-714-699-2625 / FAX: +1-847-478-2650

Northern CA Satellite
SARATOGA, CALIFORNIA 95070, U.S.A.
TEL: +1-714-699-2625 / FAX: +1-847-478-2650

Pennsylvania Service Satellite
PITTSBURGH, PENNSYLVANIA 15644, U.S.A.
TEL: +1-732-560-4500 / FAX: +1-732-560-4531

Connecticut Service Satellite
TORRINGTON, CONNECTICUT 06790, U.S.A.
TEL: +1-732-560-4500 / FAX: +1-732-560-4531

South Region Service Center
1845 SATELLITE BOULEVARD STE. 450, DULUTH, GEORGIA 30097, U.S.A.
TEL: +1-678-258-4529 / FAX: +1-678-258-4519

Texas Service Satellites
GRAPEVINE, TEXAS 76051, U.S.A.
TEL: +1-678-258-4529 / FAX: +1-678-258-4519
HOUSTON, TEXAS 77001, U.S.A.
TEL: +1-678-258-4529 / FAX: +1-678-258-4519

Tennessee Service Satellite
Nashville, Tennessee, 37201, U.S.A.
TEL: +1-678-258-4529 / FAX: +1-678-258-4519

Florida Service Satellite
WEST MELBOURNE, FLORIDA 32904, U.S.A.
TEL: +1-678-258-4529 / FAX: +1-678-258-4519

Canada Region Service Center
4299 14TH AVENUE MARKHAM, ONTARIO L3R 0J2, CANADA
TEL: +1-905-475-7728 / FAX: +1-905-475-7935

Canada Service Satellite
EDMONTON, ALBERTA T5A 0A1, CANADA
TEL: +1-905-475-7728 / FAX: +1-905-475-7935

Mexico Region Service Center
MARIANO ESCOBEDO 69 TLALNEPANTLA, 54030 EDO. DE MEXICO
TEL: +52-55-3067-7500 / FAX: +52-55-9171-7649

Monterrey Service Satellite
MONTERREY, N.L., 64720, MEXICO
TEL: +52-81-8365-4171

BRAZIL

MELCO CNC do Brasil Comércio e Serviços S.A

Brazil Region Service Center
ACESSO JOSE SARTORELLI, KM 2.1 CEP 18550-000, BOITUVA-SP, BRAZIL
TEL: +55-15-3363-9900 / FAX: +55-15-3363-9911

EUROPE

MITSUBISHI ELECTRIC EUROPE B.V.

GOTHAER STRASSE 10, 40880 RATINGEN, GERMANY
TEL: +49-2102-486-0 / FAX: +49-2102-486-5910

Germany Service Center
KURZE STRASSE, 40, 70794 FILDERSSTADT-BONLANDEN, GERMANY
TEL: +49-711-770598-123 / FAX: +49-711-770598-141

France Service Center DEPARTEMENT CONTROLE NUMERIQUE
25, BOULEVARD DES BOUVETS, 92741 NANTERRE CEDEX FRANCE
TEL: +33-1-41-02-83-13 / FAX: +33-1-49-01-07-25

France (Lyon) Service Satellite DEPARTEMENT CONTROLE NUMERIQUE
120, ALLEE JACQUES MONOD 69800 SAINT PRIEST FRANCE
TEL: +33-1-41-02-83-13 / FAX: +33-1-49-01-07-25

Italy Service Center
VIALE COLLEONI, 7 - CENTRO DIREZIONALE COLLEONI PALAZZO SIRIO INGRESSO 1
20864 AGRATE BRIANZA (MB), ITALY
TEL: +39-039-6053-342 / FAX: +39-039-6053-206

Italy (Padova) Service Satellite
VIA G. SAVELLI, 24 - 35129 PADOVA, ITALY
TEL: +39-039-6053-342 / FAX: +39-039-6053-206

U.K. Branch
TRAVELLERS LANE, HATFIELD, HERTFORDSHIRE, AL10 8XB, U.K.
TEL: +49-2102-486-0 / FAX: +49-2102-486-5910

Spain Service Center
CTRA. DE RUBI, 76-80-APDO. 420
08173 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: +48-12-630-4700 / FAX: +48-12-630-4701

Mitsubishi Electric Turkey A.Ş Ümraniye Şubesi
Turkey Service Center
ŞERİFALİ MAH. NUTUK SOK. NO.5 34775
ÜMRANIYE, İSTANBUL, TURKEY
TEL: +90-216-526-3990 / FAX: +90-216-526-3995

Czech Republic Service Center
KAFKOVA 1853/3, 702 00 OSTRAVA 2, CZECH REPUBLIC
TEL: +420-59-5691-185 / FAX: +420-59-5691-199

Russia Service Center
213, B.NOVODMITROVSKAYA STR., 14/2, 127015 MOSCOW, RUSSIA
TEL: +7-495-748-0191 / FAX: +7-495-748-0192

MITSUBISHI ELECTRIC EUROPE B.V. (SCANDINAVIA)

Sweden Service Center
HAMMARBACKEN 14 191 49 SOLLENTUNA, SWEDEN
TEL: +46-8-6251000 / FAX: +46-8-966877

Bulgaria Service Center
4 A.LYAPCHEV BOUL., POB 21, BG-1756 SOFIA, BULGARIA
TEL: +359-2-8176009 / FAX: +359-2-9744061

Ukraine (Kharkov) Service Center
APTEKARSKIY LANE 9-A, OFFICE 3, 61001 KHARKOV, UKRAINE
TEL: +380-57-732-7774 / FAX: +380-57-731-8721

Ukraine (Kiev) Service Center
4-B, M. RASKOVOYI STR., 02660 KIEV, UKRAINE
TEL: +380-44-494-3355 / FAX: +380-44-494-3366

Belarus Service Center
OFFICE 9, NEZAVISIMOSTI PR.177, 220125 MINSK, BELARUS
TEL: +375-17-393-1177 / FAX: +375-17-393-0081

South Africa Service Center
5 ALBATROSS STREET, RHODESFIELD, KEMPTON PARK 1619, GAUTENG, SOUTH AFRICA
TEL: +27-11-394-8512 / FAX: +27-11-394-8513

ASEAN**MITSUBISHI ELECTRIC ASIA PTE. LTD. (ASEAN FA CENTER)**

Singapore Service Center
307 ALEXANDRA ROAD #05-01/02 MITSUBISHI ELECTRIC BUILDING SINGAPORE 159943
TEL: +65-6473-2308 / FAX: +65-6476-7439

Malaysia (KL) Service Center
60, JALAN USJ 10/1B 47620 UEP SUBANG JAYA SELANGOR DARUL EHSAN, MALAYSIA
TEL: +60-3-5631-7605 / FAX: +60-3-5631-7636

Malaysia (Johor Baru) Service Center
17 & 17A, JALAN IMPIAN EMAS 5/5, TAMAN IMPIAN EMAS, 81300 SKUDAI, JOHOR MALAYSIA.
TEL: +60-7-557-8218 / FAX: +60-7-557-3404

Philippines Service Center
UNIT NO.411, ALABAMG CORPORATE CENTER KM 25, WEST SERVICE ROAD
SOUTH SUPERHIGHWAY, ALABAMG MUNTINLUPA METRO MANILA, PHILIPPINES 1771
TEL: +63-2-807-2416 / FAX: +63-2-807-2417

VIETNAM**MITSUBISHI ELECTRIC VIETNAM CO.,LTD**

Vietnam (Ho Chi Minh) Service Center
UNIT 01-04, 10TH FLOOR, VINCOM CENTER 72 LE THANH TON STREET, DISTRICT 1,
HO CHI MINH CITY, VIETNAM
TEL: +84-8-3910 5945 / FAX: +84-8-3910 5946

Vietnam (Hanoi) Service Satellite
6th Floor, Detech Tower, 8 Ton That Thuyet Street, My Dinh 2 Ward, Nam Tu Liem District, Hanoi, Vietnam
TEL: +84-4-3937-8075 / FAX: +84-4-3937-8076

INDONESIA**PT. MITSUBISHI ELECTRIC INDONESIA**

Indonesia Service Center (Cikarang Office)
JL.Kenari Raya Blok G2-07A Delta Silicon 5, Lippo Cikarang-Bekasi 17550, INDONESIA
TEL: +62-21-2961-7797 / FAX: +62-21-2961-7794

THAILAND**MITSUBISHI ELECTRIC FACTORY AUTOMATION (THAILAND) CO.,LTD**

Thailand Service Center
12TH FLOOR, SV.CITY BUILDING, OFFICE TOWER 1, NO. 896/19 AND 20 RAMA 3 ROAD,
KWAENG BANGPONGPANG, KHET YANNAWA, BANGKOK 10120, THAILAND
TEL: +66-2-682-6522-31 / FAX: +66-2-682-6020

INDIA**MITSUBISHI ELECTRIC INDIA PVT. LTD.**

India Service Center
2nd FLOOR, TOWER A & B, DLF CYBER GREENS, DLF CYBER CITY,
DLF PHASE-III, GURGAON 122 002, HARYANA, INDIA
TEL: +91-124-4630 300 / FAX: +91-124-4630 399
Ludhiana satellite office
Jamshedpur satellite office

India (Pune) Service Center
EMERALD HOUSE, EL-3, J-BLOCK, MIDC BHOSARI, PUNE – 411 026, MAHARASHTRA, INDIA
TEL: +91-20-2710 2000 / FAX: +91-20-2710 2100
Baroda satellite office
Mumbai satellite office

India (Bangalore) Service Center
PRESTIGE EMERALD, 6TH FLOOR, MUNICIPAL NO. 2,
LAVELLE ROAD, BANGALORE - 560 043, KAMATAKA, INDIA
TEL: +91-80-4020-1600 / FAX: +91-80-4020-1699
Chennai satellite office
Coimbatore satellite office

OCEANIA**MITSUBISHI ELECTRIC AUSTRALIA LTD.**

Australia Service Center
348 VICTORIA ROAD, RYDALMERE, N.S.W. 2116 AUSTRALIA
TEL: +61-2-9684-7269 / FAX: +61-2-9684-7245

CHINA**MITSUBISHI ELECTRIC AUTOMATION (CHINA) LTD. (CHINA FA CENTER)**

China (Shanghai) Service Center
1-3,5-10,18-23/F, NO.1386 HONG QIAO ROAD, CHANG NING QU,
SHANGHAI 200336, CHINA
TEL: +86-21-2322-3030 / FAX: +86-21-2308-3000

China (Ningbo) Service Dealer
China (Wuxi) Service Dealer
China (Jinan) Service Dealer
China (Hangzhou) Service Dealer
China (Wuhan) Service Satellite

China (Beijing) Service Center
9/F, OFFICE TOWER 1, HENDERSON CENTER, 18 JIANGUOMENNEI DAJIE,
DONGCHENG DISTRICT, BEIJING 100005, CHINA
TEL: +86-10-6518-8830 / FAX: +86-10-6518-8030
China (Beijing) Service Dealer

China (Tianjin) Service Center
UNIT 2003, TIANJIN CITY TOWER, NO 35 YOUYI ROAD, HEXI DISTRICT,
TIANJIN 300061, CHINA
TEL: +86-22-2813-1015 / FAX: +86-22-2813-1017
China (Shenyang) Service Satellite
China (Changchun) Service Satellite

China (Chengdu) Service Center
ROOM 407-408, OFFICE TOWER AT SHANGRI-LA CENTER, NO. 9 BINJIANG DONG ROAD,
JINJIANG DISTRICT, CHENGDU, SICHUAN 610021, CHINA
TEL: +86-28-8446-8030 / FAX: +86-28-8446-8630

China (Shenzhen) Service Center
ROOM 2512-2516, 25/F., GREAT CHINA INTERNATIONAL EXCHANGE SQUARE, JINTIAN RD.S.,
FUTIAN DISTRICT, SHENZHEN 518034, CHINA
TEL: +86-755-2399-8272 / FAX: +86-755-8218-4776
China (Xiamen) Service Dealer
China (Dongguan) Service Dealer

KOREA**MITSUBISHI ELECTRIC AUTOMATION KOREA CO., LTD. (KOREA FA CENTER)**

Korea Service Center
8F, Gangseo Hangang Xi-tower, 401 Yangcheon-ro, Gangseo-gu, Seoul 157-801, KOREA
TEL: +82-2-3660-9602 / FAX: +82-2-3664-8668

Korea Taegu Service Satellite
4F KT BUILDING, 1630 SANGYEOK-DONG, BUK-KU, DAEGU 702-835, KOREA
TEL: +82-53-382-7400 / FAX: +82-53-382-7411

TAIWAN**MITSUBISHI ELECTRIC TAIWAN CO., LTD. (TAIWAN FA CENTER)**

Taiwan (Taichung) Service Center (Central Area)
NO.8-1, INDUSTRIAL 16TH RD., TAICHUNG INDUSTRIAL PARK, SITUN DIST.,
TAICHUNG CITY 40768, TAIWAN R.O.C.
TEL: +886-4-2359-0688 / FAX: +886-4-2359-0689

Taiwan (Taipei) Service Center (North Area)
10F, NO.88, SEC.6, CHUNG-SHAN N. RD., SHI LIN DIST., TAIPEI CITY 11155, TAIWAN R.O.C.
TEL: +886-2-2833-5430 / FAX: +886-2-2833-5433

Taiwan (Tainan) Service Center (South Area)
11F-1., NO.30, ZHONGZHENG S. ROAD, YONGKANG DISTRICT, TAINAN CITY 71067, TAIWAN, R.O.C.
TEL: +886-6-252-5030 / FAX: +886-6-252-5031

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 2012-2015 MITSUBISHI ELECTRIC CORPORATION
ALL RIGHTS RESERVED

MITSUBISHI CNC

MITSUBISHI ELECTRIC CORPORATION

HEAD OFFICE : TOKYO BLDG.,2-7-3 MARUNOUCHI,CHIYODA-KU,TOKYO 100-8310,JAPAN

MODEL	E70 Series
MODEL CODE	100-332
Manual No.	IB-1501144