MITSUBISHI ELECTRIC

Changes for the Better

MITSUBISHI CNC

Specifications Manual

C70

IB-1500259(ENG)-F

Introduction

This manual describes the specifications of CNC C70.
To safely use this CNC module, 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
☆ : Planning

⚠ 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.
(2) In this manual, the machining center system is described as "M system" and the lathe system is described as "L system".

Precautions for Safety

Always read the specifications issued by the machine maker, this manual, related manuals and attached documents before installation, operation, programming, maintenance or inspection to ensure correct use. Understand this numerical controller, safety items and cautions before using the unit.
This manual ranks the safety precautions into "Danger", "Warning" and "Caution".

⚠ **DANGER**	When 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.

⚠ **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. ⚠ 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.

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!

本製品の取扱いについて

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

Handling of our product

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

본 제품의 취급에 대해서

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

CONTENTS

I. GENERAL SPECIFICATIONS

II Functional Specifications

C70 Series Specifications List

○: Standard △: Option ☆: Plan □: Selection

Class	C70 Series		Page
	M system	L system	
1. Control axes			**1**
1.1 Control axes			**1**
1.1.1 Number of basic control axes (NC axes)	3	2	1
1.1.2 Max. number of axes (NC axes + Spindles + PLC axes)	16	16	1
1.1.2.1 Max. number of NC axes (in total for all the part systems)	16	16	1
1.1.2.2 Max. number of spindles	7	4	1
1.1.2.3 Max. number of PLC axes	8	8	1
1.1.4 Max. number of PLC indexing axes	8	8	1
1.1.5 Number of simultaneous contouring control axes	4	4	1
1.1.6 Max. number of NC axes in a part system	8	8	1
1.2 Control part system			**2**
1.2.1 Standard number of part systems	1	1	2
1.2.2 Max. number of part systems	△7	△3	2
1.3 Control axes and operation modes			**2**
1.3.2 Memory mode	○	○	2
1.3.3 MDI mode	○	○	2
1.3.102 High-speed program server mode	△	△	2
2. Input command			**3**
2.1 Data increment			**3**
2.1.1 Least command increment			**3**
2.1.1.1 Least command increment 1μm	○	○	3
2.1.1.2 Least command increment 0.1μm	△	△	3
2.2 Unit system			4
2.2.1 Inch/Metric changeover	△	△	4
2.3 Program format			5
2.3.1 Program format			5
2.3.1.1 Format 1 for Lathe	—	○	**5**
2.3.1.2 Format 2 for Lathe	—	○	5
2.3.1.4 Format 1 for Machining center	○	—	5
2.4 Command value			**6**
2.4.1 Decimal point input I, II	○	○	6
2.4.2 Absolute/Incremental command	○	○	7
2.4.3 Diameter/Radius designation	—	○	9
3. Positioning/Interpolation			**10**
3.1 Positioning			10
3.1.1 Positioning	○	○	10
3.1.2 Unidirectional positioning	△	—	11
3.2 Linear/Circular interpolation			**12**
3.2.1 Linear interpolation	○	○	12
3.2.2 Circular interpolation (Center/Radius designation)	○	○	13
3.2.3 Helical interpolation	△	△	15
3.2.101 Hypothetical linear axis control	△	—	**17**
4. Feed			**19**
4.1 Feed rate			19
4.1.1 Rapid traverse rate (m/min)	1000	1000	19
4.1.2 Cutting feed rate (m/min)	1000	1000	**20**
4.1.3 Manual feed rate (m/min)	1000	1000	21
4.1.4 Rotary axis command speed tenfold	○	○	21
4.2 Feed rate input methods			22
4.2.1 Feed per minute	○	○	22
4.2.2 Feed per revolution	△	○	23
4.2.4 F1-digit feed	○	○	24
4.3 Override			25
4.3.1 Rapid traverse override	○	○	**25**
4.3.2 Cutting feed override	○	○	**25**
4.3.3 2nd cutting feed override	○	○	25
4.3.4 Override cancel	○	○	25
4.4 Acceleration/Deceleration			26
4.4.1 Automatic acceleration/deceleration after interpolation	○	○	26
4.4.2 Rapid traverse constant inclination acceleration/deceleration	○	○	**28**
4.5 Thread cutting			30
4.5.1 Thread cutting (Lead/Thread number designation)	△	○	30
4.5.2 Variable lead thread cutting	—	○	32
4.5.3 Synchronous tapping (with digital I/F spindle)			33
4.5.3.1 Synchronous tapping cycle	△	△	33
4.5.3.2 Pecking tapping cycle	△	—	**34**
4.5.4 Chamfering	—	○	35
4.5.8 High-speed synchronous tapping (OMR-DD)	△	△	35
4.6 Manual feed			36
4.6.1 Manual rapid traverse	○	○	36
4.6.2 Jog feed	○	○	**36**
4.6.3 Incremental feed	○	○	37
4.6.4 Handle feed	△	△	37
4.7 Dwell			38
4.7.1 Dwell (Time-based designation)	○	○	**38**

○: Standard △: Option ★: Plan □: Selection

Class	C70 Series		Page
	M system	L system	
5. Program memory/editing			**39**
5.1 Memory capacity			39
5.1.1 Memory capacity (number of programs stored)			39
5.1.1.1 15kB[40m] (64 programs)	○	○	39
5.1.1.2 30kB[80m] (128 programs)	△	△	39
5.1.1.3 60kB[160m] (200 programs)	△	△	39
5.1.1.4 125kB[320m] (200 programs)	△	△	39
5.1.1.5 230kB[600m] (400 programs)	△	△	39
5.1.1.6 500kB[1280m] (1000 programs)	△	△	**39**
5.1.1.7 1000kB[2560m] (1000 programs)	△	△	39
5.1.1.8 2000kB[5120m] (1000 programs)	△	△	39
5.2 Editing			40
5.2.1 Program editing	○	○	40
5.2.2 Background editing	○	○	41
5.2.4 Word editing	○	○	**41**
6. Operation and display			**42**
6.1 Structure of operation/display panel			**42**
6.1.2 Color display (GOT)	□	□	**42**
6.2 Operation methods and functions			43
6.2.2 Absolute value/Incremental value setting	○	○	43
6.2.3 Single-NC and multi-display unit switch	○	○	43
6.2.4 Multi-NC and common-display unit	○	○	43
6.2.5 Displayed part system switch	○	○	**43**
6.2.10 Screen saver, backlight OFF	○	○	43
6.2.15 Screen Capture	○	○	43
6.3 Display methods and contents (CNC monitor function)			44
6.3.1 Status display	○	○	**44**
6.3.2 Clock display	○	○	**44**
6.3.3 Position display	○	○	44
6.3.4 Tool compensation/Parameter	○	○	44
6.3.5 Program	○	○	45
6.3.6 Alarm diagnosis	○	○	**45**
6.3.8 Additional languages			45
6.3.8.1 Japanese	○	○	45
6.3.8.2 English	○	○	45
6.3.8.6 Spanish	△	△	45
6.3.8.7 Chinese			45
6.3.8.7.2 Simplified Chinese characters	△	△	45
6.3.8.14 Polish	△	△	45
7. Input/Output functions and devices			**46**
7.1 Input/Output data			46
7.1.1 Machining program input/output	○	○	46
7.1.2 Tool offset data input/output	○	○	46
7.1.3 Common variable input/output	○	○	46
7.1.4 Parameter input/output	○	○	46
7.1.5 History data output	○	○	**46**
8. Spindle, Tool and Miscellaneous functions			**47**
8.1 Spindle functions (S)			47
8.1.1 Spindle control functions			47
8.1.1.1 Spindle digital I/F	○	○	48
8.1.1.2 Spindle analog I/F	△(using MELSEC I/O)	△(using MELSEC I/O)	48
8.1.1.3 Coil switch	○	○	48
8.1.1.4 Automatic coil switch	○	○	48
8.1.2 S code output	○	○	48
8.1.3 Constant surface speed control	△	△	49
8.1.4 Spindle override	○	○	50
8.1.5 Multiple-spindle control			51
8.1.5.1 Multiple-spindle control I	○	○	51
8.1.6 Spindle orientation	○	○	52
8.1.7 Spindle position control (Spindle/C axis control)	△	△	52
8.1.8 Spindle synchronization			53
8.1.8.1 Spindle synchronization I	△	△	53
8.1.8.2 Spindle synchronization II	△	△	53
8.1.11 Spindle speed clamp	○	○	53
8.2 Tool functions (T)			54
8.2.1 Tool functions (T command)	○	○	54
8.3 Miscellaneous functions (M)			55
8.3.1 Miscellaneous functions	○	○	55
8.3.2 Multiple M codes in 1 block	○	○	55
8.3.3 M code independent output	○	○	55
8.3.4 Miscellaneous function finish	○	○	56
8.4 2nd miscellaneous functions (B)			57
8.4.1 2nd miscellaneous functions	○	○	**57**

○: Standard △: Option ★: Plan □: Selection

Class	C70 Series		Page
	M system	L system	
9. Tool compensation			**58**
9.1 Tool length/Tool position			58
9.1.1 Tool length compensation	○	○	58
9.2 Tool radius			61
9.2.1 Tool radius compensation	○	—	61
9.2.3 Tool nose radius compensation (G40/41/42)	—	○	63
9.2.4 Automatic decision of nose radius compensation direction (G46/40)	—	○	64
9.3 Tool offset amount			**65**
9.3.1 Number of tool offset sets			65
9.3.1.2 40 sets	○	—	65
9.3.1.3 80 sets	△	○	65
9.3.1.4 100 sets	△	—	65
9.3.1.5 200 sets	△	—	**65**
9.3.2 Offset memory			66
9.3.2.1 Tool shape/wear offset amount	○	○	**66**
10. Coordinate system			**68**
10.1 Coordinate system type and setting			68
10.1.1 Machine coordinate system	○	○	69
10.1.2 Coordinate system setting	○	○	70
10.1.3 Automatic coordinate system setting	○	○	71
10.1.4 Workpiece coordinate system selection			72
10.1.4.1 Workpiece coordinate system selection (6 sets) G54 to G59	○	○	72
10.1.4.2 Extended workpiece coordinate system selection (48 sets) G54.1P1 to P48	△	—	73
10.1.5 External workpiece coordinate offset	○	○	74
10.1.7 Local coordinate system	○	○	75
10.1.8 Coordinate system for rotary axis	○	○	76
10.1.9 Plane selection	○	○	76
10.1.10 Origin set/Origin cancel	○	○	77
10.1.11 Counter set	○	○	77
10.2 Return			78
10.2.1 Manual reference position return	○	○	78
10.2.2 Automatic 1st reference position return	○	○	79
10.2.3 2nd, 3rd, 4th reference position return	○	○	81
10.2.4 Reference position check	○	○	82
10.2.5 Absolute position detection	△	△	83
11. Operation support functions			**84**
11.1 Program control			84
11.1.1 Optional block skip	○	○	84
11.1.2 Optional block skip addition	○	○	**84**
11.1.3 Single block	○	○	85
11.2 Program test			**86**
11.2.1 Dry run	○	○	86
11.2.2 Machine lock	○	○	86
11.2.3 Miscellaneous function lock	○	○	86
11.3 Program search/start/stop			87
11.3.1 Program search	○	○	87
11.3.2 Sequence number search	○	○	**87**
11.3.5 Automatic operation start	○	○	87
11.3.6 NC reset	○	○	**87**
11.3.7 Feed hold	○	○	**88**
11.3.8 Search & Start	○	○	88
11.4 Interrupt operation			89
11.4.1 Manual interruption	○	○	89
11.4.2 Automatic operation handle interruption	○	○	**89**
11.4.3 Manual absolute switch	○	○	90
11.4.4 Thread cutting cycle retract	—	△	91
11.4.5 Tapping retract	○	○	92
11.4.6 Manual numerical value command	○	○	93
11.4.8 MDI interruption	○	○	**93**
11.4.9 Simultaneous operation of manual and automatic modes	○	○	94

○: Standard △: Option ★: Plan □: Selection

Class	C70 Series		Page
	M system	L system	
12. Program support functions			**95**
12.1 Machining method support functions			95
12.1.1 Program			95
12.1.1.1 Subprogram control	○(8 layers)	○(8 layers)	95
12.1.2 Macro program			**96**
12.1.2.1 User macro	△(4 layers)	△(4 layers)	**96**
12.1.2.3 Macro interruption	△	△	98
12.1.2.4 Variable command			99
12.1.2.4.1 100 sets	○	○	101
12.1.2.4.2 200 sets	△	△	101
12.1.2.4.3 300 sets	△	△	101
12.1.2.4.4 600 sets	△	△	101
12.1.2.4.6 (50+50× number of part systems) sets	○	○	101
12.1.2.4.7 (100+100×number of part systems) sets	△	△	101
12.1.2.4.8 (200+100×number of part systems) sets	△	△	101
12.1.2.4.9 (500+100×number of part systems) sets	△	△	101
12.1.2.101 N code macro	△	△	102
12.1.2.102 Macro interface extension (1200 sets)	△	△	102
12.1.3 Fixed cycle			103
12.1.3.1 Fixed cycle for drilling	○	○	**104**
12.1.3.3 Special fixed cycle	△	—	109
12.1.3.4 Fixed cycle for turning machining	—	○	113
12.1.3.5 Compound type fixed cycle for turning machining	—	○	118
12.1.4 Mirror image			127
12.1.4.3 Mirror image by G code	○	—	127
12.1.4.4 Mirror image for facing tool posts	—	△	128
12.1.5 Coordinate system operation			**129**
12.1.5.1 Coordinate rotation by program	△	—	**129**
12.1.6 Dimension input			130
12.1.6.1 Corner chamfering/Corner R	△	△	130
12.1.6.3 Geometric command	—	○	134
12.1.7 Axis control			**138**
12.1.7.1 Chopping			138
12.1.7.1.1 Chopping	△	△	138
12.1.7.3 Circular cutting	△	—	139
12.1.8 Multi-part system control			140
12.1.8.1 Timing synchronization between part systems	○	○	140
12.1.8.2 Start point designation timing synchronization	○	○	142
12.1.8.6 Balance cut	—	○	144
12.1.8.8 2-part system synchronous thread cutting	—	○	**145**
12.1.9 Data input by program			147
12.1.9.1 Parameter input by program	△	△	147
12.1.9.2 Compensation data input by program	△	△	148
12.1.10 Machining modal			150
12.1.10.1 Tapping mode	○	○	150
12.1.10.2 Cutting mode	○	○	150
12.2 Machining accuracy support functions			151
12.2.1 Automatic corner override	○	○	151
12.2.2 Deceleration check			**152**
12.2.2.1 Exact stop check mode	○	○	153
12.2.2.2 Exact stop check	○	○	153
12.2.2.3 Error detection	○	○	153
12.2.2.4 Programmable in-position check	○	○	154
12.3 High-speed and high-accuracy functions [kBPM:k Block per Minute]			155
12.3.1 High-speed machining mode I (G5P1)	△16.8m/min	—	155
12.3.5 High-accuracy control1(G61.1/G08)	△	—	155

○: Standard △: Option ★: Plan □: Selection

Class	C70 Series		Page
	M system	L system	
13. Machine accuracy compensation			**157**
13.1 Static accuracy compensation			157
13.1.1 Backlash compensation	○	○	157
13.1.2 Memory-type pitch error compensation	△	△	157
13.1.3 Memory-type relative position error compensation	△	△	158
13.1.4 External machine coordinate system compensation	△	△	158
13.1.5 Circular error radius compensation	△	△	**159**
13.1.6 Ball screw thermal expansion compensation	△	△	**159**
13.2 Dynamic accuracy compensation			160
13.2.1 Smooth high-gain (SHG) control	○	○	160
13.2.2 Dual feedback	○	○	161
13.2.3 Lost motion compensation	○	○	161
14. Automation support functions			**162**
14.1 Measurement			162
14.1.1 Skip			162
14.1.1.1 Skip	△	△	162
14.1.1.2 Multiple-step skip	△	△	163
14.1.2 Automatic tool length measurement	△	△	164
14.1.3 Manual tool length measurement 1	△	△	167
14.2 Tool life management			168
14.2.1 Tool life management			168
14.2.1.1 Tool life management I	△	△	168
14.2.1.2 Tool life management II	△	△	168
14.2.2 Number of tool life management sets			169
14.2.2.1 80 sets	—	△	169
14.2.2.2 100 sets	△	—	169
14.3 Others			170
14.3.1 Programmable current limitation	○	○	170
15. Safety and maintenance			**171**
15.1 Safety switches			171
15.1.1 Emergency stop	○	○	171
15.1.2 Data protection key	○	○	171
15.2 Display for ensuring safety			172
15.2.1 NC warning	○	○	172
15.2.2 NC alarm	○	○	172
15.2.3 Operation stop cause	○	○	173
15.2.4 Emergency stop cause	○	○	173
15.2.5 Thermal detection	○	○	173
15.2.6 Battery alarm/warning	○	○	173
15.3 Protection			174
15.3.1 Stroke end (Over travel)	○	○	174
15.3.2 Stored stroke limit			174
15.3.2.1 Stored stroke limit I/II	○	○	175
15.3.2.2 Stored stroke limit IB	△	△	177
15.3.2.3 Stored stroke limit IIB	△	△	177
15.3.2.4 Stored stroke limit IC	△	△	178
15.3.4 Chuck/Tailstock barrier check	—	○	179
15.3.5 Interlock	○	○	180
15.3.6 External deceleration	○	○	180
15.3.9 Door interlock			181
15.3.9.1 Door interlock I	○	○	181
15.3.9.2 Door interlock II	○	○	182
15.3.10 Parameter lock	○	○	183
15.3.11 Program protection (Edit lock B, C)	○	○	183
15.3.12 Program display lock	○	○	183
15.3.13 Safety observation	△	△	184
15.4 Maintenance and troubleshooting			185
15.4.1 Operation history	○	○	185
15.4.2 Data sampling	○	○	185
15.4.3 NC data backup	○	○	185
15.4.5 Servo automatic tuning(MS Configurator)	○	○	**186**
15.4.102 Backup	○	○	188

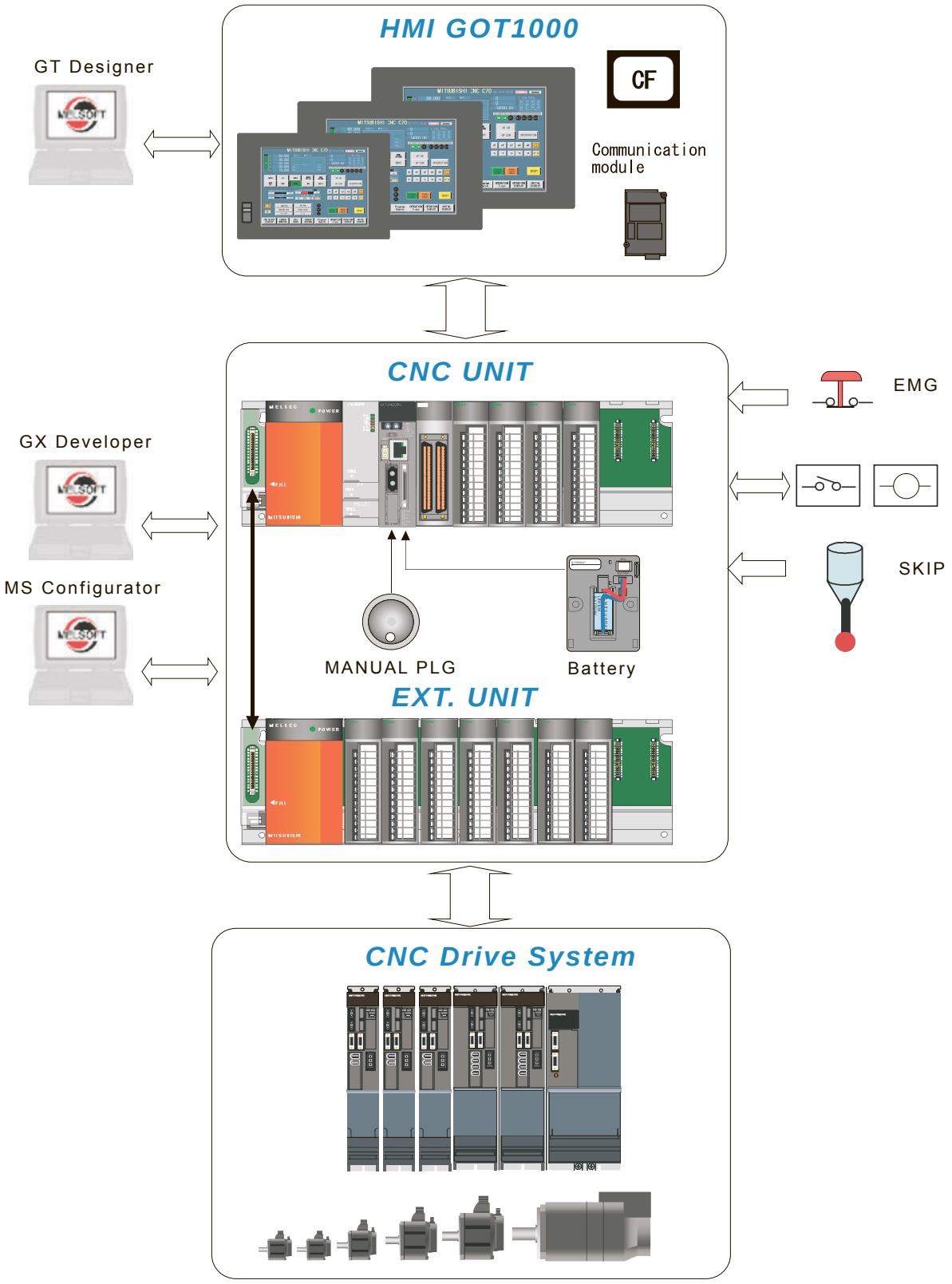
○: Standard △: Option ★: Plan □: Selection

Class	C70 Series		Page
	M system	L system	
16. Drive system			**189**
16.1 Servo/Spindle			189
16.1.1 Servo drive unit			189
16.1.1.1 MDS-D-V1/D-V2 (200V)	□	□	189
16.1.1.2 MDS-DH-V1/DH-V2 (400V)	□	□	189
16.1.1.3 MDS-D-SVJ3 (200V)	□	□	**189**
16.1.1.4 MDS-DM-V3(200V)	□	□	**189**
16.1.2 Spindle drive unit			190
16.1.2.1 MDS-D-SP/D-SP2(200V)	□	□	190
16.1.2.2 MDS-DH-SP (400V)	□	□	190
16.1.2.3 MDS-D-SPJ3 (200V)	□	□	190
16.1.3 Multi-hybrid drive unit			190
16.1.3.1 MDS-DM-SPV2/SPV3(200V)	□	□	190
16.1.4 Power supply			190
16.1.4.1 Power supply: MDS-D-CV (200V)	□	□	190
16.1.4.2 Power supply: MDS-DH-CV (400V)	□	□	190
17. Machine support functions			**191**
17.1 PLC			191
17.1.2 PLC functions			**191**
17.1.2.1 Built-in PLC basic function	△ (MELSEC)	△ (MELSEC)	**191**
17.1.2.2 NC exclusive instruction	△ (MELSEC)	△ (MELSEC)	192
17.1.2.3 Built-in PLC processing mode	○	○	192
17.1.3 PLC support functions			193
17.1.3.6 Multi-ladder program register and execution	△ (MELSEC)	△ (MELSEC)	193
17.1.3.7 Ladder program writing during RUN	△ (MELSEC)	△ (MELSEC)	193
17.1.3.8 PLC protection	△ (MELSEC)	△ (MELSEC)	193
17.1.4 Built-in PLC capacity	□30k/40k/60k/130k/260	30k/40k/60k/130k/26	193
17.1.5 Machine contact input/output I/F	△ (MELSEC)	△ (MELSEC)	193
17.1.6 Ladder monitor	○	○	**193**
17.1.7 PLC development			194
17.1.7.1 On-board development	○	○	194
17.1.7.2 MELSEC development tool (GX Developer)	○	○	194
17.1.7.3 MELSEC development tool (GX Simulator)	○	○	194
17.1.9 GOT connection			194
17.1.9.1 CPU direct connection (RS-422/RS-232C)	△ (MELSEC)	△ (MELSEC)	194
17.1.9.2 CC-Link connection (Remote device)	△ (MELSEC)	△ (MELSEC)	**194**
17.1.9.3 CC-Link connection (Intelligent terminal)	△ (MELSEC)	△ (MELSEC)	**194**
17.2 Machine construction			195
17.2.1 Servo OFF	○	○	195
17.2.2 Axis detachment	△	△	196
17.2.3 Synchronous control	△	—	197
17.2.4 Inclined axis control	—	△	200
17.2.5 Position switch	○ (16 for each part system, 16 for PLC	○ (16 for each part system, 16 for PLC	201
17.3 PLC operation			202
17.3.1 Arbitrary feed in manual mode	○	○	202
17.3.3 PLC axis control	△	△	203
17.3.5 PLC axis indexing	△	△	204
17.4 PLC interface			**206**
17.4.1 CNC control signal	○	○	206
17.4.2 CNC status signal	○	○	207
17.4.3 PLC window	△	△	209
17.4.4 External search	△	△	210
17.6 External PLC link			211
17.6.3 CC-Link (Master/Slave)	△ (MELSEC)	△ (MELSEC)	211
17.6.4 PROFIBUS-DP (Master)	△ (MELSEC)	△ (MELSEC)	211
17.6.5 DeviceNet (Master)	△ (MELSEC)	△ (MELSEC)	**211**
17.6.6 FL-net	△ (MELSEC)	△ (MELSEC)	211
17.6.7 CC-Link/LT	△ (MELSEC)	△ (MELSEC)	**211**
17.6.8 CC-Link IE	△ (MELSEC)	△ (MELSEC)	**211**
17.6.101 ASi	△ (MELSEC)	△ (MELSEC)	211
17.7 Installing S/W for machine tools			212
17.7.3 EZSocket I/F (Need separate PC S/W)	△	△	**212**
17.7.4 APLC release (Need separate PC S/W)	△	△	212
17.8 Others			212
17.8.2 CNC remote operation tool			212
17.8.2.101 Remote monitor tool	○	○	212
17.8.102 Screen library			212
17.8.102.200 Cycle monitor (Waveform display)	○	○	212

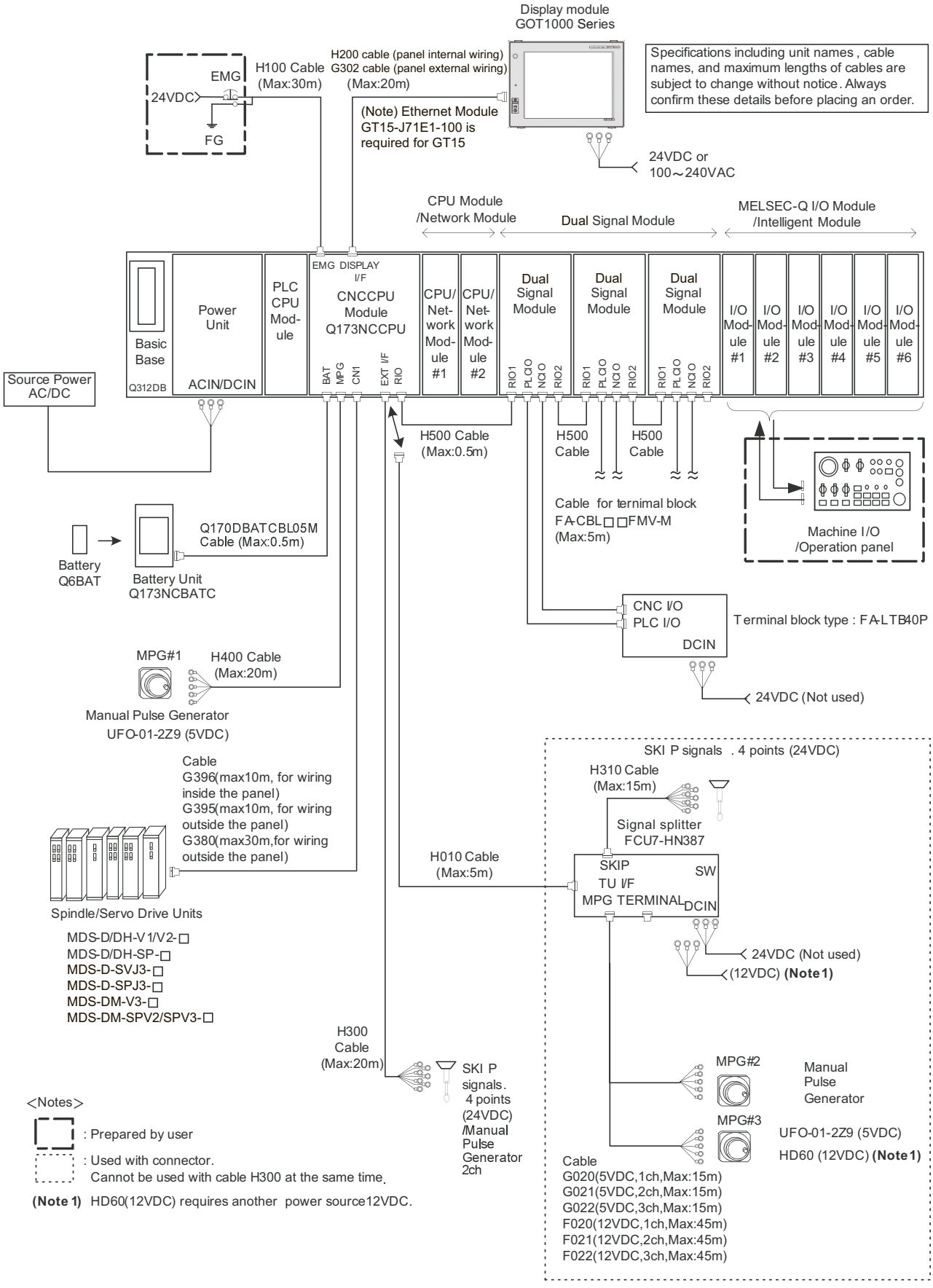
I GENERAL SPECIFICATIONS

1. System Configuration

1.1 System Basic Configuration Drawing



1.2 General Connection Diagram



1.3 Component Modules

1.3.1 CNC Control Unit

(1) Basic base

Model name	Remarks	Reference
Q38DB	8 slots	QCPU User's Manual (Hardware Design, Maintenance and Inspection) (SH(NA)-080483ENG)
Q312DB	12 slots	

(2) Power supply

Model name	Remarks	Reference
Q61P	Input power supply : 100 to 240VAC Output power supply : 5VDC Output current:6A	QCPU User's Manual (Hardware Design, Maintenance and Inspection) (SH(NA)-080483ENG)
Q63P	Input power supply: 24VDC Output power supply: 5VDC Output current: 6A	
Q64P	Input power supply: 100 to 120VAC/ 200 to 240VAC Output power supply: 5VDC Output current: 8.5A (Note) Out of production	
Q64PN	Input power supply : 100 to 240VAC Output power supply : 5VDC Output current : 8.5A	

(3) PLC CPU

Model name	Remarks	Reference
Q03UDCPU	Program capacity: 30k steps	QCPU User's Manual (Hardware Design, Maintenance and Inspection) (SH(NA)-080483ENG)
Q04UDHCPU	Program capacity: 40k steps	
Q06UDHCPU	Program capacity: 60k steps	
Q13UDHCPU	Program capacity:130k steps	
Q26UDHCPU	Program capacity:260k steps	
Q03UDECPU	Ethernet built-in type, Program capacity: 30k steps	
Q04UDEHCPU	Ethernet built-in type, Program capacity: 40k steps	
Q06UDEHCPU	Ethernet built-in type, Program capacity: 60k steps	
Q13UDEHCPU	Ethernet built-in type, Program capacity: 130k steps	
Q26UDEHCPU	Ethernet built-in type, Program capacity: 260k steps	

(4) CNC CPU module

Model name	Remarks
Q173NCCPU-S01	CNC CPU module
Battery kit	One each of following accessories are provided: Battery holder unit+Connection cable (0.5m) Q173NCBATC(Q170DBATC), Battery Q6BAT

(5) Battery holder unit

Model name	Remarks	Reference
Q173NCBATC	Battery holder unit	

(6) Input module

(a) AC

Model name	Remarks	Reference
QX10	16 points, 100 to 120VAC 8mA(100VAC, 60Hz)/7mA(100VAC, 50Hz) Response time: 20ms 16 points/common, 18-point terminal block	QCPU(Q mode) CPU Module User's Manual (Hardware) (IB(NA)-0800061)
QX28	8 points, 100 to 240VAC 17mA(200VAC, 60Hz) /14mA(200VAC, 50Hz)/8mA(100VAC, 60Hz)/ 7mA(100VAC, 50Hz) Response time: 20ms 8 points/common, 18-point terminal block	

(b) DC (positive common type)

Model name	Remarks	Reference
QX40	16 points, 24VDC, 4mA, Response time: 1/5/10/20/70ms 16 points/common, Positive common type 18-point terminal block	QCPU(Q mode) CPU Module User's Manual (Hardware) (IB(NA)-0800061)
QX40-S1	16 points, 24VDC, 6mA, Response time: 0.1/0.2/0.4/0.6/1ms 16 points/common, Positive common type 18-point terminal block	
QX41	32 points, 24VDC, 4mA, Response time: 1/5/10/20/70ms 32 points/common, Positive common type 40-pin connector	
QX41-S1	32 points, 24VDC, 4mA, Response time: 0.1/0.2/0.4/0.6/1ms 32 points/common, Positive common type 40-pin connector	
QX42	64 points, 24VDC, 4mA, Response time: 1/5/10/20/70ms 32 points/common, Positive common type 40-pin connector	
QX42-S1	64 points, 24VDC, 4mA, Response time: 0.1/0.2/0.4/0.6/1ms 32 points/common, Positive common type 40-pin connector	

(c) DC sensor

Model name	Remarks	Reference
QX70	16 points, 5/12VDC, 1.2mA(5VDC)/3.3mA(12VDC) Response time: 1/5/10/20/70ms 16 points/common, Positive/negative common type 18-point terminal block	QCPU(Q mode) CPU Module User's Manual (Hardware) (IB(NA)-0800061)
QX71	32 points, 5/12VDC, 1.2mA(5VDC)/3.3mA(12VDC) Response time: 1/5/10/20/70ms 32 points/common, Positive/negative common type 40-pin connector	
QX72	64 points, 5/12VDC, 1.2mA(5VDC)/3.3mA(12VDC) Response time: 1/5/10/20/70ms 32 points/common, Positive/negative common type 40-pin connector	

(d) DC (negative common type)

Model name	Remarks	Reference
QX80	16 points, 24VDC, 4mA Response time: 1/5/10/20/70ms 16 points/common, Negative common type 18-point terminal block	QCPU(Q mode) CPU Module User's Manual (Hardware) (IB(NA)-0800061)
QX81	32 points, 24VDC, 4mA Response time: 1/5/10/20/70ms 32 points/common, Negative common type 37-pin D sub-connector	
QX82	64 points, 24VDC, 4mA Response time: 1/5/10/20/70ms 32 points/common, Negative common type 40-pin connector	
QX82-S1	64 points, 24VDC 4mA Response time: 0.2/0.3/0.5/0.7/1.3ms 32 points/common, Negative common type 40-pin connector	

(7) Analog input module

(a) Voltage input module

Model name	Remarks	Reference
Q68ADV	8 channels, Input: -10 to 10VDC Output (resolution): 0 to 4000; -4000 to 4000; 0 to 12000; -12000 to 12000; 0 to 16000; -16000 to 16000 Conversion speed: 80μs/channel 18-point terminal block	Q64AD, Q68ADV, Q68ADI A/D Converter Module User's Manual (Hardware) (IB(NA)-0800034E)

(b) Current input module

Model name	Remarks	Reference
Q62AD-DGH	2 channels, Input: 4 to 20mADC Output (resolution): 0 to 32000; 0 to 64000 Conversion speed: 10ms/2channels 18-point terminal block, Channels are isolated, Power supply for 2-wire transmitter	Q62AD-DGH Channel Isolated High Resolution Analog-Digital Converter Module (with Signal Conditioning Function) User's Manual(Hardware) (IB(NA)-0800224E)
Q68ADI	8 channels, Input: 0 to 20mADC Output (resolution): 0 to 4000; -4000 to 4000; 0 to 12000; -12000 to 12000; 0 to 16000; -16000 to 16000 Conversion speed: 80μs/channel 18-point terminal block	Analog-Digital Converter Module User's Manual (SH(NA)-080055)

(c) Voltage/current input module

Model name	Remarks	Reference
Q64AD	4 channels, Input: -10 to 10VDC, 0 to 20mADC Output (resolution): 0 to 4000; -4000 to 4000; 0 to 12000; -12000 to 12000; 0 to 16000; -16000 to 16000 Conversion speed: 80μs/channel 18-point terminal block	Analog-Digital Converter Module User's Manual (SH(NA)-080055)
Q64AD-GH	4 channels, Input: -10 to 10VDC, 0 to 20mADC Output (resolution): 0 to 32000; -32000 to 32000; 0 to 64000; -64000 to 64000 Conversion speed: 10ms/4channels 18-point terminal block, Channels are isolated	Q64AD-GH Channel Isolated High Resolution Analog-Digital Converter Module User's Manual (Hardware) (IB(NA)-0800223E)

(8) Output module

(a) Relay

Model name	Remarks	Reference
QY10	16 points, 24VDC/240VAC, 2A/point, 8A/common Response time: 12ms 16 points/common 18-point terminal block	QCPU(Q mode) CPU Module User's Manual (Hardware) (IB(NA)-0800061)
QY18A	8 points, 24VDC/240VAC, 2A/point Response time: 12ms 18-point terminal block, All relays isolated	

(b) Triac

Model name	Remarks	Reference
QY22	16 points, 100 to 240VAC, Minimum load voltage Current: 24VAC, 100mA/100/240VAC, 25mA, OFF-time leakage current: 1.5mA(120VAC)/ 3mA(240VAC) Response time: 1ms+0.5 cycle 16 points/common, 18-point terminal block Surge killer provided	QCPU(Q mode) CPU Module User's Manual (Hardware) (IB(NA)-0800061)

(c) Transistor (sink type)

Model name	Remarks	Reference
QY40P	16 points, 12 to 24VDC OFF-time leakage current: 0.1mA Response time: 1ms, 16 points/common, Sink type 18-point terminal block, Thermal protection provided, Short circuit protection provided Surge killer provided	QCPU(Q mode) CPU Module User's Manual (Hardware) (IB(NA)-0800061)
QY41P	32 points, 12 to 24VDC OFF-time leakage current: 0,1mA Response time: 1ms, 32 points/common, Sink type 40-pin connector, Thermal protection provided Short circuit protection provided Surge killer provided	
QY42P	64 points, 12 to 24VDC OFF-time leakage current: 0.1mA Response time: 1ms, 32 points/common, Sink type 40-pin connector, Thermal protection provided Short circuit protection provided Surge killer provided	
QY50	16 points, 12 to 24VDC OFF-time leakage current: 0.1mA Response time: 1ms, 16 points/common, Sink type 18-point terminal block, Surge killer provided Fuse provided	

(d) Transistor (independent)

Model name	Remarks	Reference
QY68A	8 points, 5 to 24VDC OFF-time leakage current: 0.1mA Response time: 10ms, Sink/source type 18-point terminal block, Surge killer provided All points isolated	QCPU(Q mode) CPU Module User's Manual (Hardware) (IB(NA)-0800061)

(e) TTL CMOS

Model name	Remarks	Reference
QY70	16 points, 5 to 12VDC, Response time: 0.5ms 16 points/common, Sink type 18-point terminal block, Fuse provided	QCPU(Q mode) CPU Module User's Manual (Hardware) (IB(NA)-0800061)
QY71	32 points, 5 to 12VDC, Response time: 0.5ms 32 points/common, Sink type 40-pin connector, Fuse provided	

(f) Transistor (source type)

Model name	Remarks	Reference
QY80	16 points, 12 to 24VDC OFF-time leakage current: 0.1mA Response time: 1ms, 16 points/common Source type, 18-point terminal block Surge killer provided, Fuse provided	QCPU(Q mode) CPU Module User's Manual (Hardware) (IB(NA)-0800061)
QY81P	32 points, 12 to 24VDC OFF-time leakage current: 0.1mA Response time: 1ms, 32 points/common Source type, 37-pin D sub-connector, Thermal protection provided, Short circuit protection provided, Surge killer provided	

(9) Analog output module

(a) Voltage output module

Model name	Remarks	Reference
Q68DAVN	8 channels Input (resolution): 0 to 4000; -4000 to 4000; 0 to 12000; -12000 to 12000; -16000 to 16000 Output: -10 to 10VDC Conversion speed: 80μs/channel 18-point terminal block, Transformer insulation between power supply and output modules	Digital-Analog Converter Module User's Manual (SH(NA)-080054)

(b) Current input module

Model name	Remarks	Reference
Q68DAIN	8 channels Input (resolution): 0 to 4000; -4000 to 4000; 0 to 12000; -12000 to 12000 Output: 0 to 20mADC Conversion speed: 80μs/channel 18-point terminal block, Transformer insulation between power supply and output modules	Digital-Analog Converter Module User's Manual (SH(NA)-080054)

(c) Voltage/current output module

Model name	Remarks	Reference
Q62DAN	2 channels Input (resolution): 0 to 4000; -4000 to 4000; 0 to 12000; -12000 to 12000; -16000 to 16000 Output: -10 to 10VDC, 0 to 20mADC Conversion speed: 80μs/channel 18-point terminal block, Transformer insulation between power supply and output modules	Digital-Analog Converter Module User's Manual (Hardware) (IB(NA)-0800321E)
Q62DA-FG	2 channels Input (resolution): 0 to 12000; -12000 to 12000; -16000 to 16000 Output: -12 to 12VDC, 0 to 22mADC Conversion speed: 10ms/2channels 18-point terminal block, Channels are isolated	Q62DA-FG Channel Isolated Digital-Analog Converter Module User's Manual (Hardware) (IB(NA)-0800277E)
Q64DAN	4 channels Input (resolution): 0 to 4000; -4000 to 4000; 0 to 12000; -12000 to 12000; -16000 to 16000 Output: -10 to 10VDC, 0 to 20mADC Conversion speed: 80μs/channel 18-point terminal block, Transformer insulation between power supply and output modules	Digital-Analog Converter Module User's Manual (Hardware) (IB(NA)-0800321E)

(10) Interrupt input module

Model name	Remarks	Reference
QI60	16 points, 24VDC 4mA Response time: 0.1/0.2/0.4/0.6/1ms 16 points/common, 18-point terminal block	QCPU(Q mode) CPU Module User's Manual (Hardware) (IB(NA)-0800061)

(11) Temperature input module

(a) RTD

Model name	Remarks	Reference
Q64RD	4 channels Platinum RTD (Pt100(JIS C1604-1997, IEC 751 1983), JPt100(JISC1604-1981)) Conversion speed: 40ms/channel 18-point terminal block	RTD Input Module Channel Isolated RTD Input Module User's Manual (SH(NA)-080142)
Q64RD-G	4 channels Platinum RTD (Pt100(JIS C1604-1997, IEC 751 1983), JPt100(JISC1604-1981), Ni100Ω(DIN43760 1987)) Conversion speed: 40ms/channel 18-point terminal block, Channels are isolated	

(b) Thermocouple

Model name	Remarks	Reference
Q64TD	4 channels, Thermocouple (JIS C1602-1995) Conversion speed: 40ms/channel 18-point terminal block	Thermocouple Input Module Channel Isolated Thermocouple/Micro Voltage Input Module User's Manual(Hardware) (IB(NA)-0800155E)
Q64TDV-GH	4 channels, Thermocouple (JIS C1602-1995) Micro voltage input range: -100mV to 100mV Conversion speed: (sampling period × 3)/channel 18-point terminal block	
Q64TCTT	4 channels, Thermocouple (K, J, T, B, S, E, R, N, U, L, PLII, W5Re/W26Re) Without heater disconnection detection Sampling period: 0.5s/4channels 18-point terminal block	Temperature Control Module User's Manual (SH(NA)-080121)
Q64TCTTBW	4 channels, Thermocouple (K, J, T, B, S, E, R, N, U, L, PLII, W5Re/W26Re) With heater disconnection detection Sampling period: 0.5s/4channels 2 units of 18-point terminal block	

(c) Platinum RTD

Model name	Remarks	Reference
Q64TCRT	4 channels, Platinum RTD (Pt100, JPt100) Without heater disconnection detection Sampling period: 0.5s/4channels 18-point terminal block	Temperature Control Module User's Manual (SH(NA)-080121)
Q64TCRTBW	4 channels, Platinum RTD (Pt100, JPt100) With heater disconnection detection Sampling period: 0.5s/4channels 2 units of 18-point terminal block	

(d) Loop controller

Model name	Remarks	Reference
Q62HLC	Loop control module Thermocouple input 2ch, 5 modes of PID control Output: 4 to 20mA	Loop Control Module User's Manual (Hardware) (IB(NA)-0800319E)

(12) Channel isolated pulse input module

Model name	Remarks	Reference
QD60P8-G	8 channels 30kpps/10kpps/1kpps/100pps/50pps/ 10pps/1pps/0.1pps Count input signal: 5/12 to 24VDC	Channel Isolated Pulse Input Module User's Manual (Hardware) (IB(NA)-0800229E)

(13) High-speed counter module

Model name	Remarks	Reference
QD62	2 channels, 200/100/10kpps Count input signal: 5/12/24VDC External input: 5/12/24VDC Coincidence output: transistor (sink type) 12/24VDC, 0.5A/point, 2A/common 40-pin connector	High-Speed Counter Module User's Manual (SH(NA)-080036)
QD62D	2 channels, 500/200/100/10kpps Count input signal: EIA Standard RS-422-A (differential line driver level) External input: 5/12/24VDC Coincidence output: transistor (sink type) 12/24VDC, 0.5A/point, 2A/common 40-pin connector	
QD62E	2 channels, 200/100/10kpps Count input signal: 5/12/24VDC External input: 5/12/24VDC Coincidence output: transistor (source type) 12/24VDC, 0.1A/point, 0.4A/common 40-pin connector	

(14)Ethernet

Model name	Remarks	Reference
QJ71E71-100	10BASE-T/100BASE-TX	Q Corresponding MELSEC Communication Protocol Reference Manual (SH(NA)-080008)
QJ71E71-B2	10BASE2	
QJ71E71-B5	10BASE5	

(15)Serial communication

Model name	Remarks	Reference
QJ71C24N	RS-232 1 channel, RS-422/485 1 channel Transmission rate: 230.4kbps (Total)	Serial Communication Module User's Manual(Hardware) (SH-0800008E)
QJ71C24N-R2	RS-232 2 channels Transmission rate: 230.4kbps (Total)	
QJ71C24N-R4	RS-422/485 2 channels Transmission rate: 230.4kbps (Total)	

(16) MES interface module

Model name	Remarks	Reference
QJ71MES96	10BASE-T/100BASE-TX 1 channel (Note) MX MESInterface and CF card are separately required.	MES Interface Module User's Manual (Hardware) (IB(NA)-0800354E)

(17) MELSECNET/H

(a) SI/QSI optical interface

Model name	Remarks	Reference
QJ71LP21-25	SI/QSI/H-PCF/Broad-band H-PCF optical cable, Double loop PLC to PLC network (control/normal station)/ Remote I/O net (remote master station)	MELSECNET/H Network Module User's Manual(Hardware) (IB(NA)-0800144E)
QJ71LP21S-25	SI/QSI/H-PCF/Broad-band H-PCF optical cable, Double loop PLC to PLC network (control/normal station)/ Remote I/O net (remote master station) With external supply power	
QJ72LP25-25	SI/QSI/H-PCF/Broad-band H-PCF optical cable, Double loop Remote I/O net (remote I/O station)	

(b) GI optical interface

Model name	Remarks	Reference
QJ71LP21G	GI optical cable, Double loop PLC to PLC network (control/normal station)/ Remote I/O net (remote master station)	MELSECNET/H Network Module User's Manual(Hardware) (IB(NA)-0800144E)
QJ72LP25G	GI optical cable, Double loop Remote I/O net (remote I/O station)	MELSECNET/H Network Module User's Manual(Hardware) (IB(NA)-0800145E)

(c) Coaxial interface

Model name	Remarks	Reference
QJ71BR11	3C-2V/5C-2V coaxial cable, Single bus PLC to PLC network (control/normal station)/ Remote I/O net (remote master station)	MELSECNET/H Network Module User's Manual(Hardware) (IB(NA)-0800144E)
QJ72BR15	3C-2V/5C-2V coaxial cable, Single bus Remote I/O net (remote I/O station)	MELSECNET/H Network Module User's Manual(Hardware) (IB(NA)-0800145E)

(18) CC-Link

Model name	Remarks	Reference
QJ61BT11N	For master/local station, For QCPU Compatible with CC-Link Ver.2	CC-Link System Master/Local Module User's Manual (Hardware) (IB(NA)-0800250E)

(19) CC-Link IE controller network

Model name	Remarks	Reference
QJ71GP21-SX	CC-Link IE Optical double loop interface module (1000BASE-SX) Control/normal station	CC-Link IE Controller Network Module User's Manual (Hardware) (IB-0800364E)
QJ71GP21S-SX	CC-Link IE Optical double loop interface module (1000BASE-SX) Control/normal station With external power supply	

(20) FL-net (OPCN-2)

(a) Ver.2.00

Model name	Remarks	Reference
QJ71FL71-T-F01	10BASE-T/100BASE-TX	FL-net(OPCN-2) Interface Module User's Manual (Hardware) (IB(NA)-0800239E)
QJ71FL71-B2-F01	10BASE2	
QJ71FL71-B5-F01	10BASE5	

(b) Ver.1.00

Model name	Remarks	Reference
QJ71FL71-T	10BASE-T	FL-net(OPCN-2) Interface Module User's Manual (Hardware) (IB(NA)-0800239E)
QJ71FL71-B2	10BASE2	
QJ71FL71-B5	10BASE5	

(21) AS-i

Model name	Remarks	Reference
QJ71AS92	Master station	AS-i Master Module User's Manual (Hardware) (IB(NA)-0800225E)

(22) Extension base

Model name	Remarks	Reference
Q63B	3 slots; for mounting Q series modules including power supply module	QCPU User's Manual (Hardware Design, Maintenance and Inspection) (SH(NA)-080483ENG)
Q65B	5 slots; for mounting Q series modules including power supply module	
Q68B	8 slots; for mounting Q series modules including power supply module	
Q612B	12 slots; for mounting Q series modules including power supply module	
Q52B	2 slots; for mounting Q series modules excluding power supply module	
Q55B	5 slots; for mounting Q series modules excluding power supply module	

(23) Spring clamp terminal block

Model name	Remarks	Reference
Q6TE-18S	For 16 points I/O modules, 0.3 to 1.5mm^2 (AWG22 to 16)	Spring Clamp Terminal Block Model Q6TE-18S User's Manual (IB(NA)-0800204E)

(24) Terminal block adapter

Model name	Remarks	Reference
Q6TA32	For 32 points I/O modules, 0.5mm^2 (AWG20)	Insulation Displacement Connector for MELSEC-Q Series 32-Point I/O Module User's Manual (IB(NA)-0800228E)
Q6TA32-TOL	Q6TA32 exclusive tool	

(25) Connector/terminal block converter module

Model name	Remarks	Reference
A6TBX36-E	For negative common type input modules (standard type)	I/O module Type Building Block User's Manual (SH(NA)-080042)
A6TBX54-E	For negative common type input modules (2-wire type)	
A6TBX70	For positive common type input modules (3-wire type)	
A6TBX70-E	For negative common type input modules (3-wire type)	
A6TBY36-E	For source type output modules (standard type)	
A6TBY54-E	For source type output modules (2-wire type)	
A6TBXY36	For positive common type input modules and sink type output modules (standard type)	
A6TBXY54	For positive common type input modules and sink type output modules (2-wire type)	

(26) Cable

(a) Cables for CNC CPU

Cable type	Application	Max. length	Standard cable length (m)	Remarks
F020	Manual pulse generator: 1ch	45m	0.5, 1, 2, 3, 5, 7, 10, 15, 20	12V power supply type can be used. For Signal splitter
F021	Manual pulse generator: 2ch	45m	0.5, 1, 2, 3, 5, 7, 10, 15, 20	
F022	Manual pulse generator: 3ch	45m	0.5, 1, 2, 3, 5, 7, 10, 15, 20	
G020	Manual pulse generator: 1ch	15m	0.5, 1, 2, 3, 5, 7, 10, 15	5V power supply type can be used. For Signal splitter
G021	Manual pulse generator: 2ch	15m	0.5, 1, 2, 3, 5, 7, 10, 15	
G022	Manual pulse generator: 3ch	15m	0.5, 1, 2, 3, 5, 7, 10, 15	
G302	Display module communication (STP cross)	20m	1, 2, 3, 5, 10, 15, 20	For panel external wiring
G303	Display module communication (STP straight)	20m	1, 2, 3, 5, 10, 15, 20	For panel external wiring, when using a HUB.
G380	Optical servo communication	30m	5,10,12, 13, 15, 20, 25, 30	PCF type with outer sheath, for panel external wiring
G395	Optical servo communication	10m	1, 2, 3, 5, 7, 10	POF type with outer sheath, for panel external wiring
G396	Optical servo communication	10m	0.3, 0.5, 1, 2, 3, 5	POF type without outer sheath, for panel internal wiring
H010	Signal splitter connection	5m	0.5, 1, 2, 3, 5	
H100	Emergency stop	30m	0.5, 1, 2, 3, 5, 7, 10, 15, 20	
H200	Display module communication (UTP cross)	20m	1, 2, 3, 5, 10, 15, 20	For panel internal wiring.
H300	SKIP/manual pulse generator input	20m	0.5, 1, 2, 3, 5, 7, 10, 15, 20	
H310	SKIP connection	15m	0.5, 1, 2, 3, 5, 7, 10, 15	For Signal splitter
H400	Manual pulse generator: 1ch for 5V	20m	0.5, 1, 2, 3, 5, 7, 10, 15, 20	
H500	Dual-signal module communication	0.5m	0.1, 0.2, 0.3, 0.5	
H810	Connection cable between I/O extension connector unit (FCU7-HN831) and external Input/output unit (GT15-DIOR)	1m	0.5, 0.75, 1	

(Note) The Standard cable length column shows the lengths of the cable available from MITSUBISHI.

(b) Cable for connector and terminal block changeover unit

Model name	Remarks	Reference
AC05TB	For A6TBXY36/A6TBXY54/A6TBX70 (positive common/sink type modules), 0.5m	I/O module Type Building Block User's Manual (SH(NA)-080042)
AC10TB	For A6TBXY36/A6TBXY54/A6TBX70 (positive common/sink type modules), 1m	
AC20TB	For A6TBXY36/A6TBXY54/A6TBX70 (positive common/sink type modules), 2m	
AC30TB	For A6TBXY36/A6TBXY54/A6TBX70 (positive common/sink type modules), 3m	
AC50TB	For A6TBXY36/A6TBXY54/A6TBX70 (positive common/sink type modules), 5m	
AC80TB	For A6TBXY36/A6TBXY54/A6TBX70 (positive common/sink type modules), 8m *Common current not exceeding 0.5A	
AC100TB	For A6TBXY36/A6TBXY54/A6TBX70 (positive common/sink type modules), 10m *Common current not exceeding 0.5A	
AC05TB-E	For A6TBX36-E/A6TBY36-E/A6TBX54-E /A6TBY54-E/A6TBX70-E (negative common, source type modules), 0.5m	
AC10TB-E	For A6TBX36-E/A6TBY36-E/A6TBX54-E /A6TBY54-E/A6TBX70-E (negative common, source type modules), 1m	
AC20TB-E	For A6TBX36-E/A6TBY36-E/A6TBX54-E /A6TBY54-E/A6TBX70-E (negative common, source type modules), 2m	
AC30TB-E	For A6TBX36-E/A6TBY36-E/A6TBX54-E/ A6TBY54-E/A6TBX70-E (negative common, AC30TB-E source type modules), 3m	
AC50TB-E	For A6TBX36-E/A6TBY36-E/A6TBX54-E /A6TBY54-E/A6TBX70-E (negative common, source type modules), 5m	

(c) Cable for drive unit

Cable type	Application	Max. length	Standard cable length (m)	Remarks
CNP2E-1- ☐ M	Motor side PLG cable	30m	2, 3, 4, 5, 7, 10, 15, 20, 25, 30	
CNV22J-K1P-0.3M	Detector extension cable for HF-KP motor	0.3m	0.3	
CNV22J-K2P-0.3M	Detector extension cable for HF-KP motor	0.3m	0.3	
CNV2E-6P- ☐ M	Motor side detector cable (for A74/ A51)/ Ball screw side detector cable	30m	2, 3, 4, 5, 7, 10, 15, 20, 25, 30	
CNV2E-7P- ☐ M	Motor side detector cable (for A74/ A51)/ Ball screw side detector cable	30m	2, 3, 4, 5, 7, 10, 15, 20, 25, 30	
CNV2E-8P- ☐ M	Motor side detector cable (for A74/A51/A48)/ Ball screw side detector cable	30m	2, 3, 4, 5, 7, 10, 15, 20, 25, 30	
CNV2E-9P- ☐ M	Motor side detector cable (for A74/A51/A48)/ Ball screw side detector cable	30m	2, 3, 4, 5, 7, 10, 15, 20, 25, 30	
CNV2E-D- ☐ M	MDS-B-SD unit cable	30m	2, 3, 4, 5, 7, 10, 15, 20, 25, 30	
CNV2E-K1P- ☐ M	Detector cable for HF-KP motor	10m	2, 3, 5, 7, 10	(load side angle)
CNV2E-K2P- ☐ M	Detector cable for HF-KP motor	10m	2, 3, 5, 7, 10	(reverse load side angle)
DG21- ☐ M	Battery cable	5m	0.3, 0.5, 1, 5	(For drive unit - battery unit)
DG22- ☐ M	Battery cable	5m	0.3, 0.5, 1, 5	(For servo drive unit - servo drive unit) * This cable is required to supply the power from the battery unit to multiple drive units.
DG23- ☐ M	Battery cable	5m	0.3, 0.5, 1, 5	(For servo drive unit -battery box)
DG24- ☐ M	5V spply/DO output cable	5m	0.3, 0.5, 1, 5	(For servo drive unit -battery box)
MR-BKS1CBL ☐ M-A1-H	Brake cable for HF-KP motor	10m	2, 3, 5, 7, 10	(load side angle)
MR-BKS1CBL ☐ M-A2-H	Brake cable for HF-KP motor	10m	2, 3, 5, 7, 10	(reverse load side angle)
MR-PWS1CBL ☐ M-A1-H	Power cable for HF-KP motor	10m	2, 3, 5, 7, 10	(load side angle)
MR-PWS1CBL ☐ M-A2-H	Power cable for HF-KP motor	10m	2, 3, 5, 7, 10	(reverse load side angle)
SH21	Power supply communication cable	30m	0.35, 0.5, 1, 2, 3, 5, 10, 15, 20, 30	

(Note) The Standard cable length column shows the lengths of the cable available from MITSUBISHI.

(27) Relay terminal unit

(a) Unit

Model name	Remarks	Reference
A6TE2-16SRN	40 pin connector For 24VDC Transistor output unit (sink type module)	Relay Terminal Module User's Manual (Hardware) A6TE2-16SRN IB-68932

(b) Cable

Model name	Remarks	Reference
AC06TE	For A6TE2-16SRN 0.6m	Relay Terminal Module User's Manual (Hardware) A6TE2-16SRN IB-68932
AC10TE	For A6TE2-16SRN 1m	
AC30TE	For A6TE2-16SRN 3m	
AC50TE	For A6TE2-16SRN 5m	
AC100TE	For A6TE2-16SRN 10m	

(28) Extension cable

Model name	Remarks	Reference
QC05B	0.45m cable	QCPU User's Manual (Hardware Design, Maintenance and Inspection) (SH(NA)-080483ENG)
QC06B	0.6m cable	
QC12B	1.2m cable	
QC30B	3m cable	
QC50B	5m cable	
QC100B	10m cable	

(29) Connector

Model name	Remarks	Reference
A6CON1	Soldering type 32 point-connector (40-pin connector)	I/O module Type Building Block User's Manual (SH(NA)-080042)
A6CON2	Crimp-contact type 32 point-connector (40-pin connector)	
A6CON3	Flat cable pressure displacement type 32-point connector (40-pin connector)	
A6CON4	Soldering type 32 point-connector (40-pin connector; two-way cable can be mounted)	
A6CON1E	Soldering type 32 point-connector (37-pin D sub-connector)	
A6CON2E	Crimp-contact type 32 point-connector (37-pin D sub-connector)	
A6CON3E	Flat cable pressure displacement type 32-point connector (37-pin D sub-connector)	

1.3.2 GOT

1.3.2.1 GT16

(1) GOT

(a) GT1695M

Model name	Remarks	Reference
GT1695M-XTBA	15.0 type, XGA [1024 × 768 dots] TFT color liquid crystal display (High intensity and wide angle view), 65536 colors <Multi-media and video/RGB supported> 100-240VAC, built-in flash memory 15MB	GT16 General Description (IB-0800434)
GT1695M-XTBD	15.0 type, XGA [1024 × 768 dots] TFT color liquid crystal display (High intensity and wide angle view), 65536 colors <Multi-media and video/RGB supported> 24VDC, built-in flash memory 15MB	

(b) GT1685M

Model name	Remarks	Reference
GT1685M-STBA	12.1 type, SVGA [800 × 600 dots] TFT color liquid crystal display (High intensity and wide angle view), 65536 colors <Multi-media and video/RGB supported> 100-240VAC, built-in flash memory 15MB	GT16 General Description (IB-0800434)
GT1685M-STBD	12.1 type, SVGA [800 × 600 dots] TFT color liquid crystal display (High intensity and wide angle view), 65536 colors <Multi-media and video/RGB supported> 24VDC, built-in flash memory 15MB	

(c) GT1675M

Model name	Remarks	Reference
GT1675M-STBA	10.4 type, SVGA [800 × 600 dots] TFT color liquid crystal display (High intensity and wide angle view), 65536 colors <Multi media and video/RGB supported> 100-240VAC, built-in flash memory 15MB	GT16 General Description (IB-0800434)
GT1675M-STBD	10.4 type, SVGA [800 × 600 dots] TFT color liquid crystal display (High intensity and wide angle view), 65536 colors <Multi media and video/RGB supportedd> 24VDC, built-in flash memory 15MB	

(d) GT1665M

Model name	Remarks	Reference
GT1665M-STBA	8.4 type, SVGA [800 × 600 dots] TFT color liquid crystal display (High intensity and wide angle view), 65536 colors <Multi media and video/RGB supported> 100-240VAC, built-in flash memory 15MB	GT16 General Description (IB-0800434)
GT1665M-STBD	8.4 type, SVGA [800 × 600 dots] TFT color liquid crystal display (High intensity and wide angle view), 65536 colors <Multi media and video/RGB supported> 24VDC, built-in flash memory 15MB	

(2) Option function board

Model name	Remarks	Reference
GT16-MESB	For MES interface function	GT16 MES Interface Function Board User's Manual (IB-0800427E)

(3) Protection sheet

Model name	Remarks	Reference
GT16-90PSCB	Protection sheet for 15.0 type (Clear, 5 sheets)	GT16 Protective Sheet User's Manual (IB-0800427E)
GT16-90PSGB	Protection sheet for 15.0 type (Anti-glare, 5 sheets)	
GT16-80PSCB	Protection sheet for 12.1 type (Clear, 5 sheets)	
GT16-80PSGB	Protection sheet for 12.1 type (Anti-glare, 5 sheets)	
GT16-70PSCB	Protection sheet for 10.4 type (Clear, 5 sheets)	
GT16-70PSGB	Protection sheet for 10.4 type (Anti-glare, 5 sheets)	
GT16-60PSCB	Protection sheet for 8.4 type (Clear, 5 sheets)	
GT16-60PSGB	Protection sheet for 8.4 type (Anti-glare, 5 sheets)	

1.3.2.2 GT15

(1) GOT

(a) GT1595

Model name	Remarks	Reference
GT1595-XTBA	15.0 type, XGA [1024×768 dots] TFT color liquid crystal display (High intensity and wide angle view), 65536 colors, 100-240VAC, built-in flash memory 9MB	GT15 General Description (IB(NA)-0800322E)
GT1595-XTBD	15.0 type, XGA [1024×768 dots] TFT color liquid crystal display (High intensity and wide angle view), 65536 colors, 24VDC, built-in flash memory 9MB	

(b) GT1585

Model name	Remarks	Reference
GT1585V-STBA	12.1 type, SVGA [800×600 dots] TFT color liquid crystal display (High intensity and wide angle view), 65536 colors <Video/RGB supported> 100-240VAC, built-in flash memory 9MB	GT15 General Description (IB(NA)-0800322E)
GT1585V-STBD	12.1 type, SVGA [800×600 dots] TFT color liquid crystal display (High intensity and wide angle view), 65536 colors <Video/RGB supported> 24VDC, built-in flash memory 9MB	
GT1585-STBA	12.1 type, SVGA [800×600 dots] TFT color liquid crystal display (High intensity and wide angle view), 65536 colors 100-240VAC, built-in flash memory 9MB	
GT1585-STBD	12.1 type, SVGA [800×600 dots] TFT color liquid crystal display (High intensity and wide angle view), 65536 colors 24VDC, built-in flash memory 9MB	

(c) GT1575

Model name	Remarks	Reference
GT1575V-STBA	10.4 type, SVGA [800×600 dots] TFT color liquid crystal display (High intensity and wide angle view), 65536 colors <Video/RGB supported> 100-240VAC, built-in flash memory 9MB	GT15 General Description (IB(NA)-0800322E)
GT1575V-STBD	10.4 type, SVGA [800×600 dots] TFT color liquid crystal display (High intensity and wide angle view), 65536 colors <Video/RGB supported> 24VDC, built-in flash memory 9MB	
GT1575-STBA	10.4 type, SVGA [800×600 dots] TFT color liquid crystal display (High intensity and wide angle view), 65536 colors 100-240VAC, built-in flash memory 9MB	
GT1575-STBD	10.4 type, SVGA [800×600 dots] TFT color liquid crystal display (High intensity and wide angle view), 65536 colors 24VDC, built-in flash memory 9MB	

(2) Communication unit

(a) Ethernet communication unit

Model name	Remarks	Reference
GT15-J71E71-100	Ethernet (100Base-TX/10Base-T) unit Necessary for connecting to Q173NCCPU	GT15 Ethernet communication unit User's Manual (IB(NA)-0800314E)

(3) Option function board

Model name	Remarks	Reference
GT15-QFNB	Select either of these models when using GOT options (MELSEC-Q/QnA circuit monitor functions).	GT15 Option Function Board/ Option Function Board with Add-on Memory User's Manual (IB(NA)-0800301E)
GT15-QFNB16M		
GT15-QFNB32M		
GT15-QFNB48M		
GT15-MESB48M		

(4) Protection sheet

Model name	Remarks	Reference
GT15-90PSCB	Protection sheet for 15.0 type (Clear/5 sheets)	GT15 Protective Sheet User's Manual (IB(NA)-0800295)
GT15-80PSCB	Protection sheet for 12.1 type (Clear/5 sheets)	
GT15-70PSCB	Protection sheet for 10.4 type (Clear/5 sheets)	

1.3.2.3 Option

(1) CF card extension interface

Model name	Remarks	Reference
GT15-CFEX-C08-SET	CF card extension interface (front)	GT15 CF card extension unit User's Manual (IB(NA)-0800367E)

(2) External input/output unit

Model name	Remarks	Reference
GT15-DIOR	(Input)16 points/Output for scan 8 points 24VDC about 4mA (Output)16 points+1 point (RUN output) 24VDC 0.1A/point (Negative common input/source type output)	GT15 External I/O Unit (Negative Common Input/ Source Type Output) User's Manual (IB(NA)-0800425)
GT15-DIO	(Input)16 points/Output for scan 8 points 24VDC about 4mA (Output)16 points+1 point (RUN output) 24VDC 0.1A/point (Positive common input/sink type output)	GT15 External I/O Unit (Positive Common Input/Sink Type Output) User's Manual (IB(NA)-0800382)

1.3.3 Peripheral Device

(1) Signal splitter

Model name	Remarks
FCU7-HN387	Option (Manual pulse generator is required for 2 or 3 axes specifications)

(2) Manual pulse generator

Model name	Remarks
UFO-01-2Z9	5V specifications
HD60	12V specifications, for connection to operation panel I/O module 12V power supply is separately required.

(3) I/O extension connector unit

Model name	Remarks
FCU7-HN831	Point extension unit of external input/output unit GT15-DIOR

1.3.4 Dual Signal Module

(1) Dual signal module

Model name	Remarks
Q173SXY	I/O duplication monitoring module (Maximum 3 modules)
Q173SXY-2	I/O duplication monitoring module (High speed type) (Maximum 3 modules)

(2) Terminal block

Model name	Remarks
FA-LTB40P	Terminal block converter module (Arrangement : MITSUBISHI ELECTRIC ENGINEERING COMPANY LIMITED)

(3) Cable

Model name	Remarks
FA-CBL□□FMV-M	Cable for terminal block converter module (Cable length□□ = 05:0.5m, 10:1m, 20:2m, 30:3m, 50:5m) (Arrangement : MITSUBISHI ELECTRIC ENGINEERING COMPANY LIMITED)

2. General Specifications

For the specifications of GOT, CNC servo/spindle drive unit and I/O module, refer to the manuals written in “System Configuration: Component Modules".

2.1 Installation Environment Conditions

C70, which is an open equipment, must be installed within a sealed metal control panel (IP54 or higher). C70 must also be used and stored under the conditions listed in the table of specifications below.

Item	Specification					
Operating ambient Temperature	0 to 55°C (32 to 131°F)					
Storage ambient Temperature	-25 to 75°C (-13 to 167°F)					
Operating ambient Humidity	5 to 95%RH non-condensing					
Storage ambient Humidity	5 to 95%RH non-condensing					
Vibration resistance			Frequency	Acceleration	Amplitude	Sweep count
	Under intermittent vibration		10 to 57Hz	-	0.075mm	10 times each in X, Y, Z directions (For 80 min.)
			57to 150Hz	9.8m/s^2	-	
	Under continuous vibration		10 to 57Hz	-	0.035mm	
			57 to 150Hz	4.9m/s^2	-	
Shock resistance	147m/s^2 , 3 times in each of 3 directions X, Y, Z					
Operating ambience	No corrosive gases nor inflammable gases					
Operating altitude	2000m(6561.68ft.) or less **(Note 3)**					
Installation location	Inside control panel					
Overvoltage category **(Note 1)**	II or less					
Pollution level **(Note 2)**	2 or less					

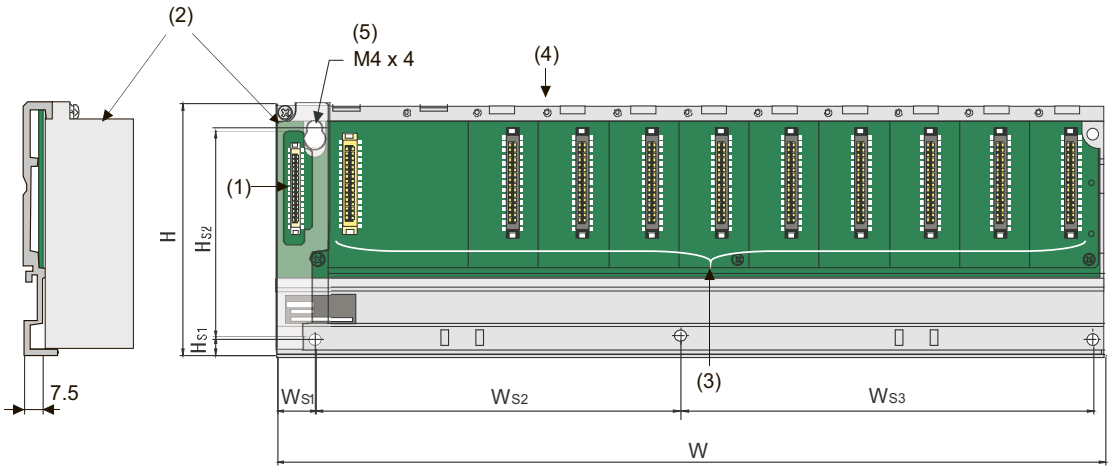
(Note 1) This indicates the section of the power supply to which the equipment is assumed to be connected between the public electrical power distribution network and the machinery within premises. Category II applies to equipment for which electrical power is supplied from fixed facilities. The surge voltage withstand level for up to the rated voltage of 300V is 2500V.

(Note 2) This index indicates the degree to which conductive material is generated in terms of the environment in which the equipment is used. Pollution level 2 is when only non-conductive pollution occurs. A temporary conductivity caused by condensing must be expected occasionally.

(Note 3) Do not use or store C70 under pressure higher than the atmospheric pressure of altitude 0m. Doing so can cause an operation failure.

(Note 4) The following environment conditions are also required for the layout design.
- No large amount of conductible dust, iron filings, oil mist, salt, or organic solvents
- No direct sunlight
- No strong electrical or magnetic fields
- No direct vibrations nor shocks on C70

2.2 Base Unit



	Q38DB	Q312DB	Q63B	Q65B	Q68B	Q612B
W	328	439	189	245	328	439
W_{S1}	15.5					
W_{S2}	170±0.3	170±0.3	167±0.3	222.5±0.3	190±0.3	190±0.3
W_{S3}	138±0.3	249±0.3			116±0.3	227±0.3
H	98					
H_{S1}	7					
H_{S2}	80±0.3					

[mm]

No.	Name	Application
(1)	Extension cable connector	Connector to which the extension cables are connected for sending and receiving signals from the extension base unit.
(2)	Base cover	Protective cover of extension cable connector. Before an extension cable is connected, the area of the base cover surrounded by the groove under the word "OUT" on the base cover must be removed with a tool such as nippers.
(3)	Module connector	Connector for installing the Q series power supply module, CPU module, I/O modules, and intelligent function module. To the connectors located in the spare space where these modules are not installed, attach the supplied connector cover or the blank cover module QG60 to prevent entry of dirt.
(4)	Module fixing screw hole	Screw hole for fixing the module to the base unit. Screw size: M3×12
(5)	Base fixing hole	Hole for fixing this base unit onto the panel of the control panel. (for M4 screw)

(Note) DIN rail installation is not available when installing the CNC CPU module onto the basic base unit.
The installation may cause the module's malfunction due to vibration.

2.3 Power Supply

C70 uses Q61P (100-240VAC input, 5VDC 6A output), Q63P (24VDC input, 5VDC 6A output) or Q64P (100-120VAC/200-240VAC input, 5VDC 8.5A output) or Q64PN(100-240VAC input, 5VDC 8.5A output).

Specifications

Item		Q61P
Base loading position		Q series power supply module loading slot
Applicable base unit		Q38DB, Q312DB, Q63B, Q65B, Q68B, Q612B
Input power supply		100 - 240VAC+10%-15% (85 - 264VAC)
Input frequency		50/60Hz ± 5%
Input voltage distortion factor		5% or less
Max. input apparent power		130VA
Inrush current		20A 8ms or less*4
Rated output current	5VDC	6A
	24VDC	-
Overcurrent protection*1	5VDC	6.6A or more
	24VDC	-
Overvoltage protection**2	5VDC	5.5 to 6.5V
Efficiency		70% or more
Permissible instantaneous power off time*3		20ms or less
Dielectric withstand voltage		Across inputs/LG and outputs/FG 2830VAC rms/3 cycles (Altitude: 2000m)
IInsulation resistance		Across inputs and outputs (LG and FG separated), across inputs for LG/FG, across outputs for LG/FG 10MΩ or more by insulation resistance tester (500VDC)
Noise immunity		By noise simulator of 1500Vp-p noise voltage, 1μs noise width and 25 to 60Hz noise frequency Noise voltage IEC61000-4-4, 2kV
Operation display		LED display (Normal: ON(Green), Error: OFF)
Fuse		Built-in (Unchangeable by user)
Contact output section	Application	ERR contact
	Rated switching voltage/ current	24VDC, 0.5A
	Minimum switching load	5VDC, 1mA
	Response time	OFF to ON:10ms or less, ON to OFF:12ms or less
	Life time	Mechanical: 20 million times or more Electrical: 100 thousand times or more at rated switching voltage/current
	Surge suppressor	None
	Fuse	None
Terminal screw size		M3.5 screw
Applicable size of wire		0.75 to 2mm^2
Applicable crimping terminal		RAV1.25-3.5, RAV2-3.5
Applicable tightening torque		0.66 to 0.89N m
Mass [kg]		0.4

Item		Q63P	Q64P	Q64PN
Base loading position		Q series power supply module loading slot		
Applicable base unit		Q38DB, Q312DB, Q63B, Q65B, Q68B, Q612B		
Input power supply		24VDC+30%-35% (15.6 to 31.2VDC)	100 to 120VAC+10%-15% /200 to 240VAC+10%-15% (85 to 132VAC/170 to 264VAC)	100 to 240VAC+10%-15% (85 to 264VAC)
Input frequency		-	50/60Hz ± 5%	
Input voltage distortion factor		-	5% or less	
Max. input apparent power		45W	160VA	
Input current		at 24VDC input: 1.82A or less at 15.6VDC input: 2.8A or less	at 100VAC input: 1.3A or less at 200VAC input: 0.75A or less	
Repetitive peak current		-	4A or less	
Inrush current		100A 1ms or less (at 24VDC input)	20A 8ms or less*4	
Rated output current	5VDC	6A	8.5A	
	24VDC	-	-	
Overcurrent protection*1	5VDC	6.6A or more	9.9A or more	
	24VDC	-	-	
Overvoltage protection**2	5VDC	5.5 to 6.5V		
Efficiency		70% or more		
Permissible instantaneous power off time*3		10ms or less (at 24VDC input)	20ms or less	
Dielectric withstand voltage		500VAC across primary and 5VDC	Across inputs/LG and outputs/FG 2,830VAC rms/3 cycles (Altitude: 2,000m (6,561.68ft.))	
Insulation resistance		10MΩ or more (measured with an insulation resistance tester)	Input and LG batched, output and FG batched, batch input-LG, batch output-FG 10MΩ or m ore by insulation resistance tester (500VDC)	
Noise immunity		By noise simulator of 500Vp-p noise voltage, 1μs noise width and 25 to 60Hz noise frequency	By noise simulator of 1,500Vp-p noise voltage, 1μs noise width and 25 to 60Hz noise frequency Noise voltage IEC61000-4-4, 2kV	
Operation display		LED display (Normal: ON(Green), Error: OFF)	LED display (Normal: ON(Green), Error: OFF)*5	LED display (Normal: ON(Green), Error: OFF)
Fuse		Built-in (Unchangeable by user)		
Contact output section	Application	ERR contact		
	Rated switching voltage/current	24VDC, 0.5A		
	Minimum switching load	5VDC, 1mA		
	Response time	OFF to ON: 10ms or less , ON to OFF: 12ms or less		
	Life time	Mechanical: 20 million times or m ore Electrical: 100 thousand times or more at rated switching voltage/current		
	Surge suppressor	Note		
	Fuse	None		
Terminal screw s ize		M3.5 screw		
Applicable size of wire		0.75 to 2mm^2		
Applicable crimping terminal		RAV1.25-3.5, RAV2-3.5		
Applicable tightening torque		0.66 to 0.89 N m		
Mass [kg]		0.33	0.4	0.47

*1: Overcurrent protection
The overcurrent protection device shuts off the 5V, 24VDC circuit and stops the system if the current flowing in the circuit exceeds the specified value.
The LED of the power supply module is turned off or lights up in dim green when voltage is lowered.
If this device is activated, switch the input power supply off and eliminate the cause such as insufficient current capacity or short. Then, a few minutes later, switch it on to restart the system.
The initial start for the system takes place when the current value becomes normal.

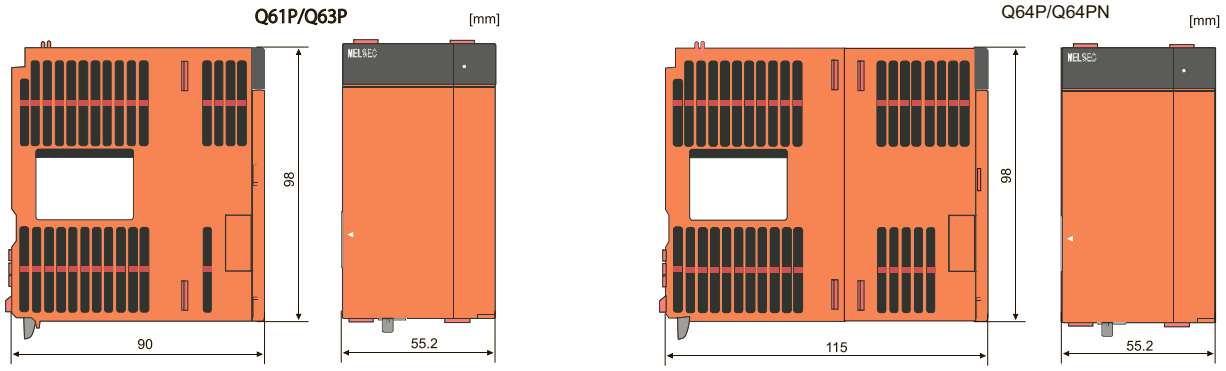
*2: Overvoltage protection
The overvoltage protection device shuts off the 5VDC circuit and stops the system if a voltage of 5.5VDC or more is applied to the circuit.
When this device is activated, the power supply module LED is switched OFF.
To restart the system, switch the input power OFF, then a few minutes later ON.
The initial start for the system will take place.
The power supply module must be changed if the system is not booted and the LED remains OFF.

*3: Permissible instantaneous power off time
(1) For AC input power supply
(a) An instantaneous power failure lasting less than 20ms will cause AC down to be detected, but operation will continue.
(b) An instantaneous power failure lasting in excess of 20ms may cause the operation to continue or initial start to take place depending on the power supply load.
Further, when the AC supply of the AC input module is the same as that of the power supply module, it prevents the sensor connected to the AC input module, which is ON at power-off, from turning OFF by switching off the power supply.
However, if only the AC input module is connected to the AC line, which is connected to the power supply, detection of the AC down for the power supply module may be delayed by the capacitor in the AC input module. Thus, connect a load of approx. 30mA per AC input module to the AC line.
(2) For DC input power supply
(a) An instantaneous power failure lasting less than 10ms* will cause 24VDC down to be detected, but operation will continue.
(b) An instantaneous power failure lasting in excess of 10ms* may cause the operation to continue or initial start to take place depending on the power supply load.
*: This is for a 24VDC input. This is 10ms or less for less than 24VDC.

*4: Inrush current
When power is switched on again immediately (within 5 seconds) after power-off, an inrush current of more than the specified value (2ms or less) may flow. Reapply power 5 seconds after power-off.
When selecting a fuse and breaker in the external circuit, take account of the blow out, detection characteristics and above matters.

*5: Operation indication
During the operation, do not allow the input voltage to change from 200VAC level (170 to 264VAC) to 100VAC level (85 to 132VAC).
(If changed, the POWER LED of the module turns off and the system operation stops.)

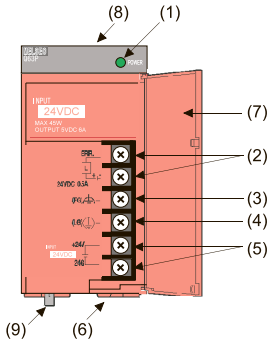
Outline dimension



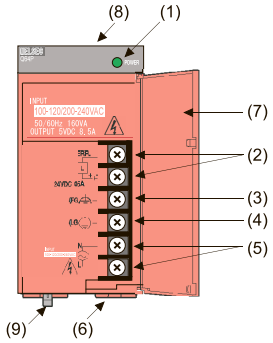
Names of parts

The following shows the names of the parts of each power module.

- Q63P (24VDC input, 5VDC 6A output)
- Q64P (100 to 120VAC/200 to 240VAC input, 5VDC 8.5A output)
- Q61P (100 - 240VAC input, 5VDC 6A output)
- Q64PN(100-240VAC input, 5VDC 8.5A output)



Q63P



Q61P/Q64P/Q64PN

(1) POWER LED

Q61P/Q64P/Q64PN

ON(green): Normal (5VDC output, instantaneous power failure within 20ms)

OFF:

- The power supply module is out of order while AC power supply is ON. (5VDC error, internal circuit failure, blown fuse)
- Over current protection or over voltage protection operated.
- AC power supply is not ON
- Power failure (including an instantaneous power failure of more than 20ms)

Q63P

ON(green): Normal (5VDC output, instantaneous power failure within 10ms)

OFF:

- The power supply module is out of order while DC power supply is ON. (5VDC error, internal circuit failure, blown fuse)
- Over current protection or over voltage protection operated.
- DC power supply is not ON
- Power failure (including an instantaneous power failure of more than 10ms)

(2) $\overline{\text{ERR}}$ terminal

Q61P/Q64P/Q64PN

- Turned ON when the whole system operates normally.
- This terminal turns OFF (opens) when the AC power is not input, a stop error (including a reset) occurs in the CPU module, or the fuse is blown.
- In a Multiple CPU system configuration, turned OFF when a stop error occurs in any of the CPU modules.
- Normally OFF when loaded in an extension base unit.

Q63P

- Turned ON when the whole system operates normally.
- This terminal turns OFF (opens) when the DC power is not input, a stop error (including a reset) occurs in the CPU module, or the fuse is blown.
- In a Multiple CPU system configuration, turned OFF when a stop error occurs in any of the CPU modules.
- Normally OFF when loaded in an extension base unit.

(3) FG terminal

Ground terminal connected to the shield pattern of the printed circuit board.

(4) LG terminal

- Grounding for the power supply filter.
- This terminal has potential of 1/2 of the input voltage for AC input (Q61P, Q64P and Q64PN).
- This is also a protective earth terminal (PE).

(5) Power input terminals

- Power input terminals connected to a power supply of 100VAC or 200VAC. (Q64P and Q64PN)
- Power input terminals connected to a power supply of 24VDC. (Q63P)
- Power input terminals connected to a power supply of 100-200VAC.(Q61P)

(6) Terminal screw

M3.5 × 7 screw

(7) Terminal cover

Protective cover of the terminal block

(8) Module fixing screw hole

Used to fix the module to the base unit.

M3 × 12 screw (user-prepared) (Tightening torque: 0.36 to 0.48 N m)

(9) Module loading lever

Used to load the module into the base unit.

(Note1) Q63P is dedicated for inputting a voltage of 24VDC. Q63P may break down unless connected to 24VDC for inputting or with reversed polarity.

(Note2) Q64P automatically switches the input range 100/200VAC.
Therefore, it is not compatible with the intermediate voltage (133 to 169VAC).
The CPU module may not work normally if the above intermediate voltage is applied.
Also note that Q64P may break down when connected to the power supply whose voltage or frequency is out of the specifications.

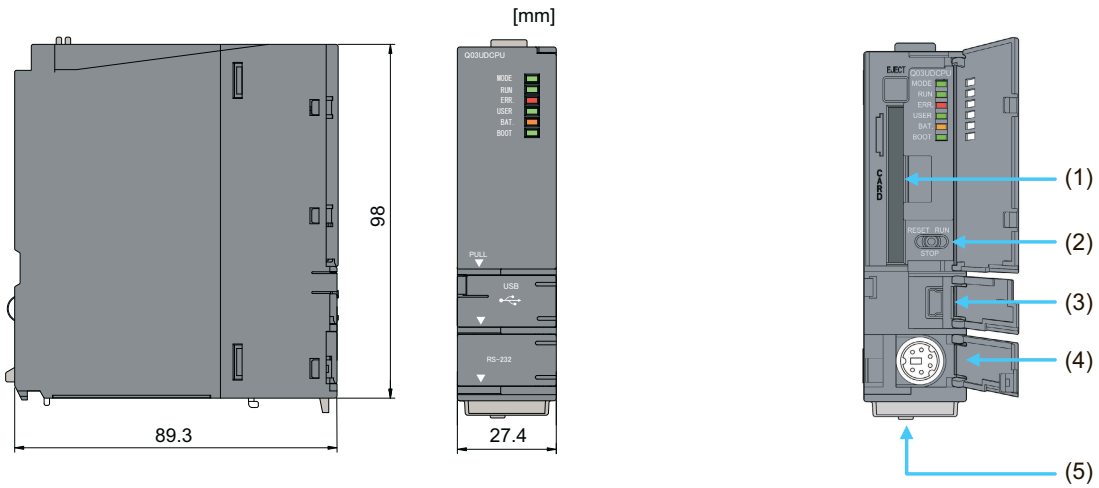
(Note3) Ensure that the earth terminals LG and FG are grounded. (Ground resistance: 100 or less)
Since the LG terminals have potential of 1/2 input voltage, the operator may receive an electric shock when touching metal parts.

(Note4) When Q61P, Q63P, Q64P or Q64PN is loaded on the extension base unit, a system error cannot be detected by the $\overline{\text{ERR}}$ terminal. ($\overline{\text{ERR}}$ terminal is always OFF.)

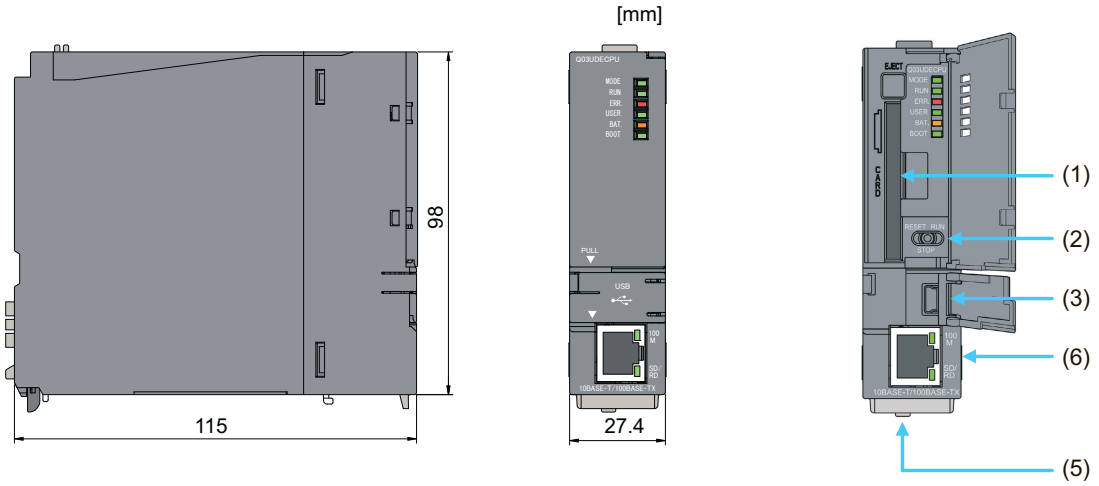
2.4 PLC CPU

For the further details than the following descriptions, refer to "QCPU User's Manual (Hardware Design, Maintenance and Inspection)" (SH(NA)-080483ENG).

Dimension and Names of parts



[Q03UDCPU / Q04UDHCPU / Q06UDHCPU / Q13UDHCPU / Q26UDHCPU]



[Q03UDECPU / Q04UDEHCPU / Q06UDEHCPU / Q13UDEHCPU / Q26UDEHCPU]

(1) CARD : Memory card slot for C70
(2) SW : RUN, STOP and RESET switches
(3) USB : USB connector for the connection of a tool
(4) RS232 : RS-232C connector for the connection of a tool
(5) BAT : Battery

PLC CPU module type	Frequency of battery usage*1	Power-ON time ratio*2	Life time of the battery		
			Guaranteed value*3 (70 ℃)	Actual service value*4 (40 ℃)	Backup time after alarm*5
Q03UD(E)CPU	1	0%	30,100hr	43,800hr	600hr
		30%	43,000hr	43,800hr	600hr
		50%	43,800hr	43,800hr	600hr
		70%	43,800hr	43,800hr	600hr
		100%	43,800hr	43,800hr	600hr
	2	0%	25,300hr	43,800hr	600hr
		30%	36,100hr	43,800hr	600hr
		50%	43,800hr	43,800hr	600hr
		70%	43,800hr	43,800hr	600hr
		100%	43,800hr	43,800hr	600hr
Q04UD(E)HCPU	1	0%	30,100hr	43,800hr	600hr
		30%	43,000hr	43,800hr	600hr
		50%	43,800hr	43,800hr	600hr
		70%	43,800hr	43,800hr	600hr
		100%	43,800hr	43,800hr	600hr
	2	0%	4,300hr	32,100hr	384hr
		30%	6,100hr	43,800hr	384hr
		50%	8,600hr	43,800hr	384hr
		70%	14,300hr	43,800hr	384hr
		100%	43,800hr	43,800hr	384hr
Q06UD(E)HCPU	1	0%	25,300hr	43,800hr	600hr
		30%	36,100hr	43,800hr	600hr
		50%	43,800hr	43,800hr	600hr
		70%	43,800hr	43,800hr	600hr
		100%	43,800hr	43,800hr	600hr
	2	0%	4,200hr	32,100hr	384hr
		30%	6,000hr	43,800hr	384hr
		50%	8,400hr	43,800hr	384hr
		70%	14,000hr	43,800hr	384hr
		100%	43,800hr	43,800hr	384hr
	3	0%	2,300hr	19,200hr	192hr
		30%	3,200hr	27,400hr	192hr
		50%	4,600hr	38,400hr	192hr
		70%	7,600hr	43,800hr	192hr
		100%	43,800hr	43,800hr	192hr

PLC CPU module type	Frequency of battery usage*1	Power-ON time ratio*2	Life time of the battery		
			Guaranteed value*3 (70 ℃)	Actual service value*4 (40 ℃)	Backup time after alarm*5
Q13UD(E)HCPU Q20UD(E)HCPU Q26UD(E)HCPU	1	0%	22,600hr	43,800hr	600hr
		30%	32,200hr	43,800hr	600hr
		50%	43,800hr	43,800hr	600hr
		70%	43,800hr	43,800hr	600hr
		100%	43,800hr	43,800hr	600hr
	2	0%	4,100hr	26,200hr	384hr
		30%	5,800hr	37,400hr	384hr
		50%	8,200hr	43,800hr	384hr
		70%	13,600hr	43,800hr	384hr
		100%	43,800hr	43,800hr	384hr
	3	0%	2,300hr	18,600hr	192hr
		30%	3,200hr	26,500hr	192hr
		50%	4,600hr	37,200hr	192hr
		70%	7,600hr	43,800hr	192hr
		100%	43,800hr	43,800hr	192hr
	4	0%	1,500hr	13,800hr	144hr
		30%	2,100hr	19,700hr	144hr
		50%	3,000hr	27,600hr	144hr
		70%	5,000hr	43,800hr	144hr
		100%	43,800hr	43,800hr	144hr

*1: The frequency of battery usage indicated battery consumption of PLC CPU. (Target CPU modules for Q03UDCPU, Q04UDHCPU, and Q06UDHCPU are the first 5 digits of the serial No. is “10012” or later.) The bigger the frequency of battery usage is, the higher amount of battery per unit time is consumed. The frequency of battery usage depends on the elements (a) and (b). The following table shows the relationship between the combination pattern of (a) and (b) and the frequency of battery usage.

Elemets to decide how much battery is used		Frequency of battery usage
(a) Battery long-life function (Note)	(b) State of a file storage during standard RAM Size of a register file during RAM (SR) < Unit: word	
With setting	-	1
Without setting	No file register or 0k<SR <= 128k	2
	128k<SR <= 384k	3
	384k<SR	4

(Note) Refer to the following manual for battery long-life function.
QnUCPU User’s Manual (Function Explanation, Program Fundamentals) SH-080807(ENG)

*2: The power-on time ratio indicates the ratio of PLC power-on time to one day (24 hours). (When the total power-on time is 12 hours and the total power-off time is 12 hours, the power-on time ratio is 50%.)

*3: The guaranteed value; equivalent to the total power failure time that is calculated based on the characteristics value of the memory (SRAM) supplied by the manufacturer and under the storage ambient temperature range of -25 to 75 (operating ambient temperature of 0 to 55).

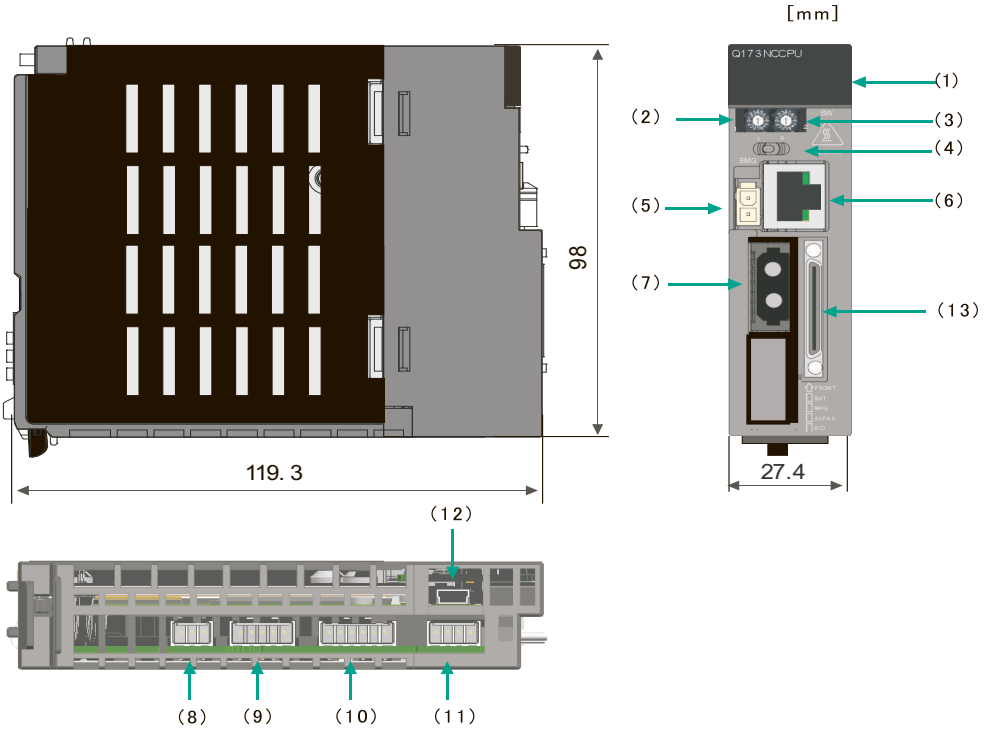
*4: The actual service value; equivalent to the total power failure time that is calculated based on the measured value and under the storage ambient temperature of 40. This value is intended for reference only, as it varies with characteristics of the memory.

*5: In the following status, the backup time after power OFF is 3 minutes.
-The battery connector is disconnected.
-The lead wire of the battery is broken.

(6) Ethernet: Ethernet connector

2.5 CNC CPU Module

Dimension and Names of parts



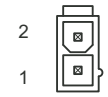
(1) LED : Display of state/alarm code (with 3 digits)

(2) SW1 : Rotary switch for maintenance (usually set to “0”)

(3) SW2 : Rotary switch for maintenance (usually set to “0”)

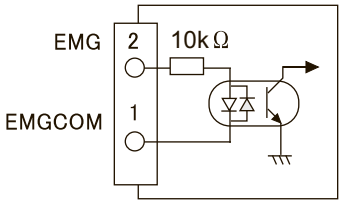
(4) SW : (Not used)

(5) EMG : Connector for the emergency stop signal input



5566-02A-210
(MOLEX)

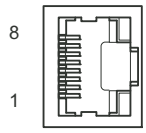
1	IN	EMGCOM
2	IN	EMG



Input type : Current sinking/sourcing
Insulation method : Photocoupler insulation
Input voltage : 24VDC (+10/-15%, ripple ratio within 5%)
OFF voltage/current : 17.5VDC or more / 3.0mA or less
ON voltage/current : 1.8VDC or less / 0.18mA or less
Input resistance : Approximate 10k
Response time (OFF -> ON or ON -> OFF): 1ms
Applicable size of wire : 0.3mm^2

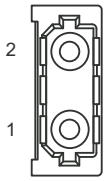
(Note) The emergency stop function suits "Stop category 1" of European safety standard “EN60204-1".

(6) DISPLAY I/F : Connector for display (GOT)



1	OUT	TD+
2	OUT	TD−
3	IN	RD+
4		CMTR
5		CMTR
6	IN	RD−
7		CMTT
8		CMTT

(7) CN1 : Connector for servo/spindle drive unit



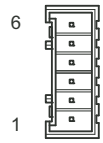
1	IN	RD
2	OUT	TD

(8) RIO : Connector for Dual signal module

1	IN/OUT	RXTXH
2	IN/OUT	RXTXL
3		SG(V)

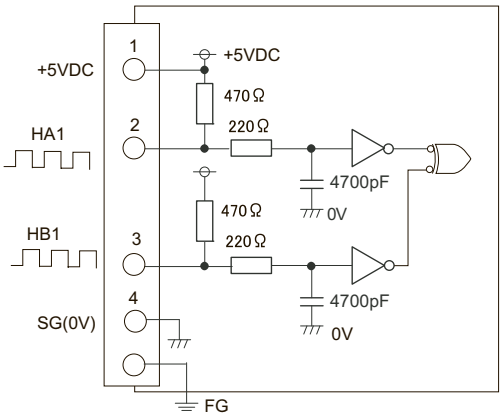
(9) AC FAIL : (Not used)

(10) MPG : Connector for 5V manual pulse generator

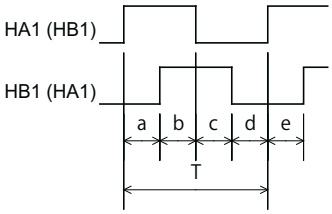


53426-0610
(MOLEX)

1	OUT	5VDC
2	IN	HA1
3	IN	HB1
4	OUT	SG(0V)
5		---
6		FG

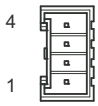


Input pulse signal type : 90° phase difference between HA1 and HB1.
Max. input pulse frequency : 100kHz
Number of pulses per rotation : 100pulse/rev
Input signal voltage : H level 3.5V to 5.25V, L level 0V to 0.5V
Power voltage for pulse generators : 5VDC ± 10%
Max. output current for pulse generators: :100mA



a.b.c.d.e: HA1 or HB1 rising edge (falling edge) phase difference = T/4 ± T/10
T: Ha1 or HB1 phase cycle (Min. 10 μ s)

(11) BAT : Connector for battery

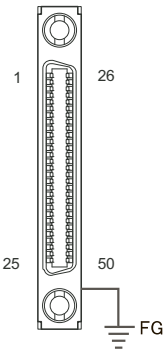


53426-0410
(MOLEX)

1	IN	FG
2		
3	IN	BAT(-)
4	IN	BAT(+)

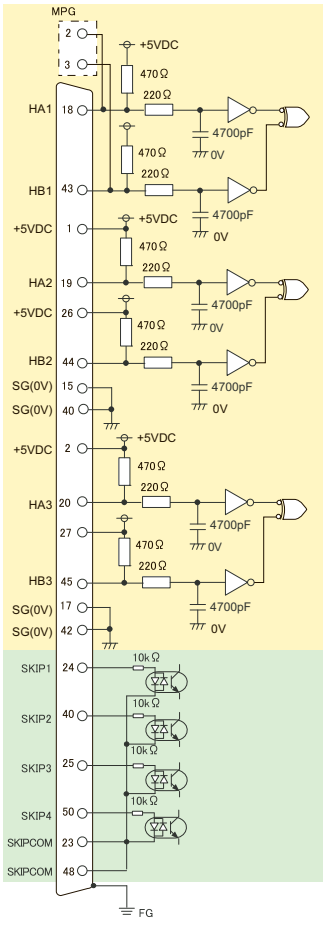
(12) Service : Connector for MITSUBISHI's servicing (Do not use)

(13) EXT I/F : Connector for the expansion connection of skip signal/ 5V manual pulse generator

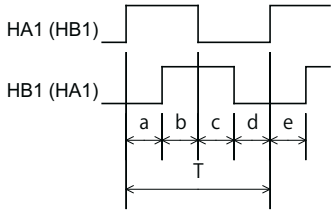


HDR-EC50LFDT1-SDL+
(HONDA)

1	OUT	5V	26	OUT	5V
2	OUT	5V	27	OUT	5V
3	OUT	SG(0V)	28	OUT	SG(0V)
4	(Reserve)		29	OUT	SG(0V)
|			30	(Reserve)	
11			| 36		
12	OUT	SG(0V)	37	OUT	SG(0V)
13		(Reserve)	38		(Reserve)
14		(Reserve)	39		(Reserve)
15	OUT	SG(0V)	40	OUT	SG(0V)
16		(Reserve)	41		(Reserve)
17	OUT	SG(0V)	42	OUT	SG(0V)
18	IN	HA1	43	IN	HB1
19	IN	HA2	44	IN	HB2
20	IN	HA3	45	IN	HB3
21		(Reserve)	46		(Reserve)
22		(Reserve)	47		(Reserve)
23	IN	SKIPCOM	48	IN	SKIPCOM
24	IN	SKIP1	49	IN	SKIP2
25	IN	SKIP3	50	IN	SKIP4

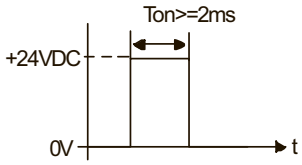


---Manual pulse generator I/F specification----
Input pulse signal type: 90° phase difference between HA1 and HB1.
Max. input pulse frequency : 100kHz
Number of pulses per rotation: 100pulse/rev
Input signal voltage : H level 3.5V to 5.25V, L level 0V to 0.5V
Output power voltage : +5VDC -10% -10%
Max. output current : 100mA
(Note) The connector MPG and EXT I/F have input pins for HA1 and HB1. Use either of the connectors.



a.b.c.d.e: HA1 or HB1 rising edge (falling edge) phase difference = T/4 ± T/10
T: HA1 or HB1 cycle (Min. 10 μs)

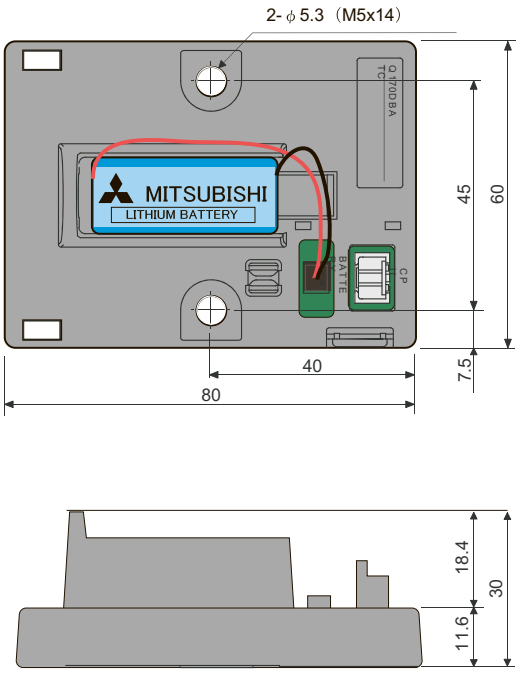
---SKIP I/F specification---
Input ON voltage : 18V or more to 25.2V or less
Input ON current : 6mA or more
Input OFF voltage : 4V or less
Input OFF current : 2mA or less
Input signal holding time (Ton) : 2ms or more
Internal response time :0.08ms or less



(Note) NC recognizes input signals of 2ms or more as the valid skip signals. If machine contacts (relay, etc.) are used, malfunctions will occur due to chattering. Use semiconductor contacts (transistor, etc.).

2.6 Battery Box for CNC CPU (Q173NCCPU)

Dimension



Life time of the battery

CNC CPU module type	Power-on time ratio*1	Life time of the battery		
		Guaranteed value*2 (75C°)	Actual service value*3 (40C°)	Backup time after alarm*4
Q173NCCPU	0%	20,000hr	43,800hr	90hr (after SM51 or SM52 ON)
	30%	27,000hr		
	50%	31,000hr		
	70%	36,000hr		
	100%	43,800hr		

*1: The power-on time ratio indicates the ratio of C70 power-on time to one day (24 hours). (When the total power-on time is 12 hours and the total power-off time is 12 hours, the power-on time ratio is 50%.)

*2: The guaranteed value; equivalent to the total power failure time that is calculated based on the characteristics value of the memory (SRAM) supplied by the manufacturer and under the storage ambient temperature range of -25 to 75 (operating ambient temperature of 0 to 55).

*3: The actual service value; equivalent to the total power failure time that is calculated based on the measured value and under the storage ambient temperature of 40. This value is intended for reference only, as it varies with characteristics of the memory.

*4: In the following status, the backup time after power OFF is 3 minutes.
- The battery connector is disconnected.
- The lead wire of the battery is broken.

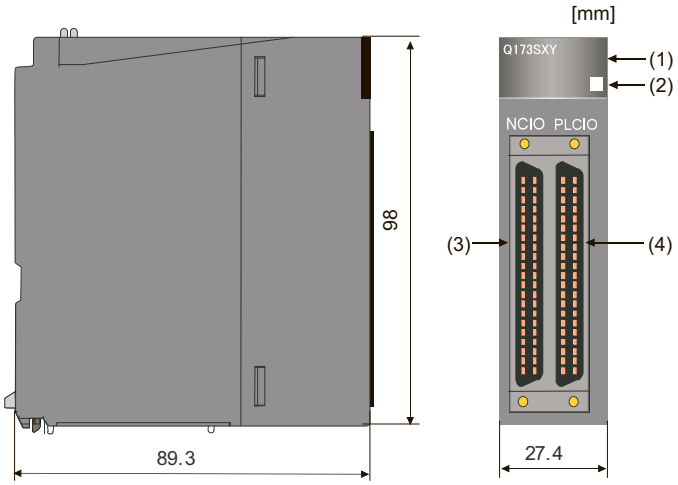
*5: The battery should be changed after 5 years of use even an alarm has not occurred.

2.7 Dual Signal Module

Use the dual signal module within the following specifications.

Items	Specifications	
	Q173SXY	Q173SXY-2
Number of input points	32 points x 2 systems (32 points for PLC CPU control + 32 points for CNC CPU control, 20 points x 2 systems for safety input, 12 points x 2 systems for feedback input for output)	
Input insulation method	Photocoupler insulation	
Rated input voltage	24VDC (+20/-15%, ripple ratio within 5%)	
Rated input current	Approximate 4mA	
Input derating	Refer to the derating figure	
ON voltage / ON current	19V or more / 3mA or more	
OFF voltage / OFF current	11V or less / 1.7mA or less	
Input resistance	Approximate 5.6kΩ	
Input response time	PLC CPU control input: 10ms (default value for digital filter) CNC CPU control input: 10ms (for CR filter)	PLC CPU control input: 10ms (default value for digital filter) CNC CPU control input: 2ms (for CR filter)
Input common method	32 points/common (Common terminal 1A01, 1A02, 2A01, 2A02) (NCIO connector and PLCIO connector have each different common)	
Input type	Type 1, Current sinking	
Number of output points	12 points x 2 systems (12 points for PLC CPU control + 12 points for CNC CPU control)	
Output insulation method	Photocoupler insulation	
Rated load voltage	24VDC(+20/-15%)	
Maximum load current	(0.1A x 8 points, 0.2A x 4 points) x 2 systems Common current: 1.6A or less for each connector	
Utilisation category	DC12/DC13	
Maximum rush current	0.7A,10ms or less (1.4A, 10ms or less for 0.2A output pin)	
OFF-time leakage current	0.1mA or less	
ON-time maximum voltage drop	0.1VDC(TYP.)0.1A, 0.2VDC(MAX.)0.1A	
Output response time	1ms or less (at rated load and resistance load)	
Output common method	12 points/common (Common terminal 1B01, 1B02, 2B01, 2B02) (NCIO connector and PLCIO connector have each different common)	
Output	Current sourcing	
Surge killer	Zener diode	
Fuse	Not provided	
External power supply	24VDC (+20/-15%, ripple ratio within 5%)	
Protection	Provided (thermal protection and short circuit protection) Thermal protection works for each 2 points. Short circuit protection works for each 1 point. (1 to 3A/point)	
Withstand voltage	560VAC rms/3cycles (at 2000m elevation)	
Insulation resistance	10MΩ or more (measured with an insulation resistance tester)	
Noise withstand level	Simulator noise 500Vp-p, Noise width 1μs measured with a noise simulator with noise frequency 25 to 60Hz	
	First transient noise IEC61000-4-4: 1kV	
Protection degree	IP2X	
Number of I/O occupational points	32 points (with I/O assignments as 32 points I/O mixed unit)	
Operation display	ON display (LED) and 32 input points display for PLC CPU control	
External connection method	40-pin connector	
Applicable size of wire	0.3mm^2 (for A6CON1 and A6CON4)	
Connector for external wiring	A6CON1, A6CON2, A6CON3, A6CON4 (sold separately)	
Terminal block changeover unit	FA-LTB40P (Cable FA-CBL ☐☐ FMV-M)	
5VDC internal power dissipation	200mA (TYP. when all points are ON)	
Mass	0.15kg	

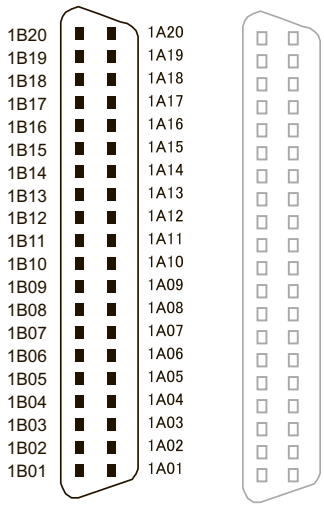
Names of parts



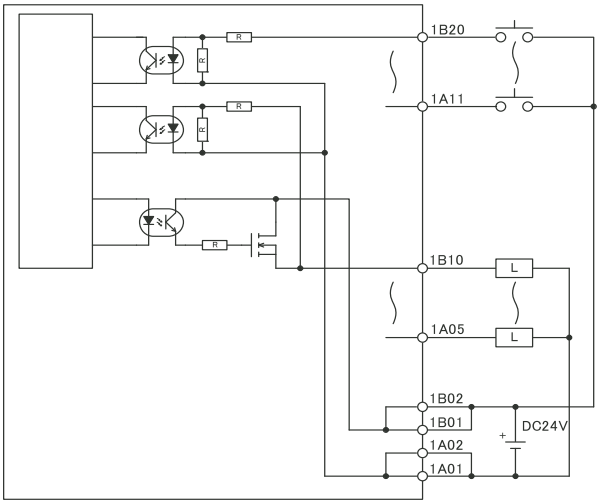
(1) LED:
Shows the input signal state of PLCIO.
(2)Module No. sticker:
Module Nos. (1 to 3) should be written on this sticker when multiple dual signal modules are mounted.

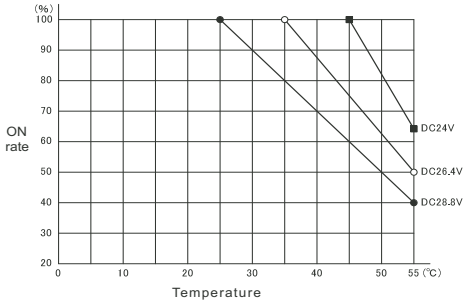
(3) NCIO:

Connector for I/O signals controlled by NCCPU (Q173NCCPU)



1B20	IN	NC-X00	1A20	IN	NC-X10
1B19	IN	NC-X01	1A19	IN	NC-X11
1B18	IN	NC-X02	1A18	IN	NC-X12
1B17	IN	NC-X03	1A17	IN	NC-X13
1B16	IN	NC-X04	1A16	IN	NC-X14
1B15	IN	NC-X05	1A15	IN	NC-X15
1B14	IN	NC-X06	1A14	IN	NC-X16
1B13	IN	NC-X07	1A13	IN	NC-X17
1B12	IN	NC-X08	1A12	IN	NC-X18
1B11	IN	NC-X09	1A11	IN	NC-X19
1B10(*)	IN/OUT	NC-Y0A/X0A	1A10(*)	IN/OUT	NC-Y1A/X1A
1B09(*)	IN/OUT	NC-Y0B/X0B	1A09(*)	IN/OUT	NC-Y1B/X1B
1B08	IN/OUT	NC-Y0C/X0C	1A08	IN/OUT	NC-Y1C/X1C
1B07	IN/OUT	NC-Y0D/X0D	1A07	IN/OUT	NC-Y1D/X1D
1B06	IN/OUT	NC-Y0E/X0E	1A06	IN/OUT	NC-Y1E/X1E
1B05	IN/OUT	NC-Y0F/X0F	1A05	IN/OUT	NC-Y1F/X1F
1B04		---	1A04		---
1B03		---	1A03		---
1B02		24VDC(COM1)	1A02		0V(COM2)
1B01		24VDC(COM1)	1A01		0V(COM2)





(Note 1) Output pins with (*) allow 0.2A output. Other pins have 0.1A output.

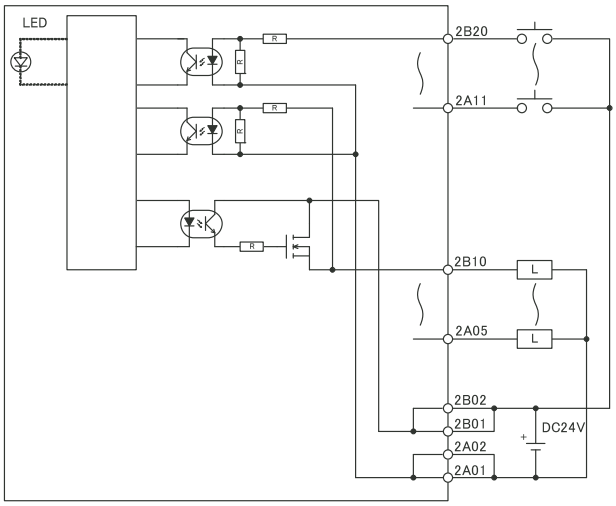
(Note 2) Pins with signal names "NC-Y0A" and "NC-X0A" are the output signals controlled by CNC CPU. When any of the signals is output to Y0A, the signal is input to X0A as a feedback signal.

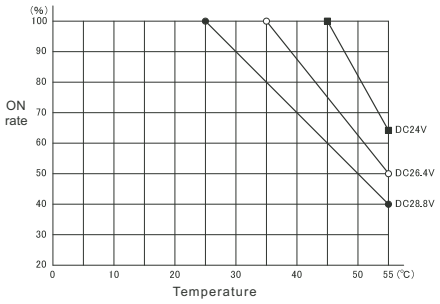
(Note 3) The device Nos. written above are for the assignment on hardware. These Nos. are different from the device Nos. to be actually used.

(4) PLCIO: Connector for I/O signals controlled by PLCCPU (QnUDHCPU).

2B20	IN	PLC-X00
2B19	IN	PLC-X01
2B18	IN	PLC-X02
2B17	IN	PLC-X03
2B16	IN	PLC-X04
2B15	IN	PLC-X05
2B14	IN	PLC-X06
2B13	IN	PLC-X07
2B12	IN	PLC-X08
2B11	IN	PLC-X09
2B10(*)	IN/OUT	PLC-Y0A/X0A
2B09(*)	IN/OUT	PLC-Y0B/X0B
2B08	IN/OUT	PLC-Y0C/X0C
2B07	IN/OUT	PLC-Y0D/X0D
2B06	IN/OUT	PLC-Y0E/X0E
2B05	IN/OUT	PLC-Y0F/X0F
2B04		---
2B03		---
2B02		24VDC(COM1)
2B01		24VDC(COM1)

2A20	IN	PLC-X10
2A19	IN	PLC-X11
2A18	IN	PLC-X12
2A17	IN	PLC-X13
2A16	IN	PLC-X14
2A15	IN	PLC-X15
2A14	IN	PLC-X16
2A13	IN	PLC-X17
2A12	IN	PLC-X18
2A11	IN	PLC-X19
2A10(*)	IN/OUT	PLC-Y1A/X1A
2A09(*)	IN/OUT	PLC-Y1B/X1B
2A08	IN/OUT	PLC-Y1C/X1C
2A07	IN/OUT	PLC-Y1D/X1D
2A06	IN/OUT	PLC-Y1E/X1E
2A05	IN/OUT	PLC-Y1F/X1F
2A04		---
2A03		---
2A02		0V(COM2)
2A01		0V(COM2)





(Note 1) Output pins with (*) allow 0.2A output. Other pins have 0.1A output.
(Note 2) The device Nos. written above are for the assignment on hardware. These Nos. are different from the device Nos. to be actually used.

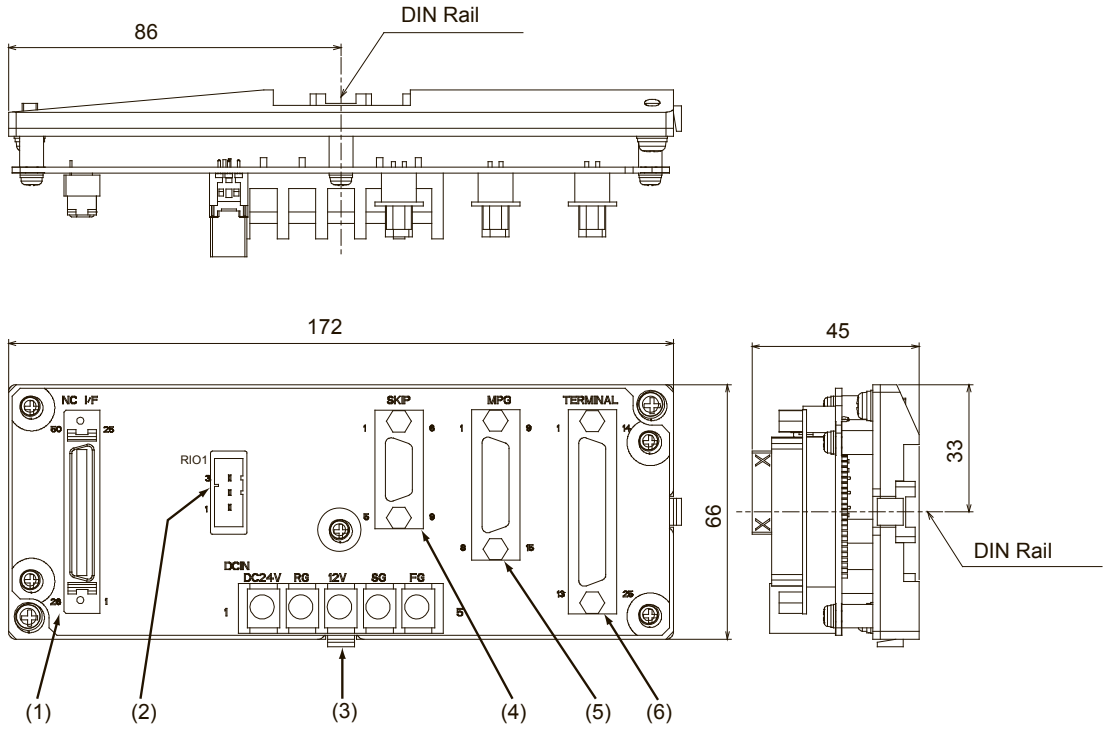
Cable side connector type

Connector type	Pressure displacement type	Crimp-contact type	Soldering type
Connector	FCN-367J040-AU/F	FCN-363J040	FCN-361J040-AU
Contact	-	AWG#24 to #28: FCN-363J-AU AWG#22 to #26: FCN-363J-AU/S	-
Case	-	FCN-360C040-B FCN-360C040-D (Wide-mouthed type) FCN-360C040-E (Long screw type)	
		-	FCN-360C040-H/E (Side-mouthed type) FCN-360C040-J1 (Sloped-mouth cover) FCN-360C040-J2 (Thin sloped-mouth cover)
Manufacturer	FUJITSU Component		

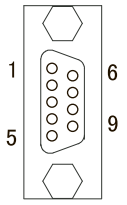
2.8 Signal Splitter

(Note) Signal splitter allows DIN rail installation only.

Dimension and Names of parts



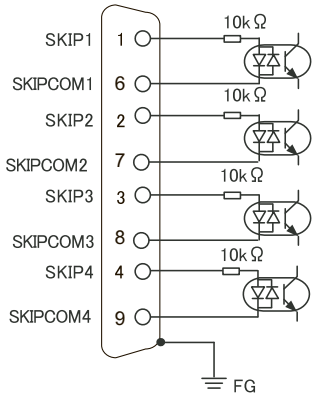
(1) NC I/F : Connector for CNC CPU
(2) RIO1 : (Not used)
(3) DCIN : Terminal block for power supply (Used for the 12V power supply type manual pulse generator)
(4) SKIP : Connector for skip signal



D−SUB 9pin

1	IN	SKIP1
2	IN	SKIP2
3	IN	SKIP3
4	IN	SKIP4
5		---

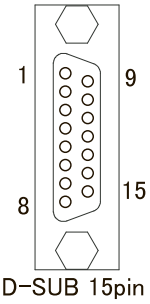
6	IN	COM1
7	IN	COM2
8	IN	COM3
9	IN	COM4



---SKIP I/F specification---
Input ON voltage : 18V or more to 25.2V or less
Input ON current : 6mA or more
Input OFF voltage : 4V or less
Input OFF current : 2mA or less
Input signal holding time (Ton) : 2ms or more
Internal response time : 0.08ms or less

(Note) NC recognizes input signals of 2ms or more as the valid skip signals. If machine contacts (relay, etc.) are used, malfunctions will occur due to chattering. Use semiconductor contacts (transistor, etc.).

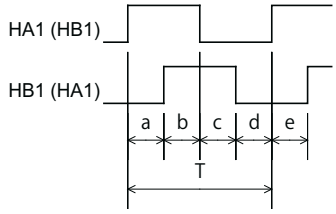
(5) MPG : 5V/12V Connector for manual pulse generator



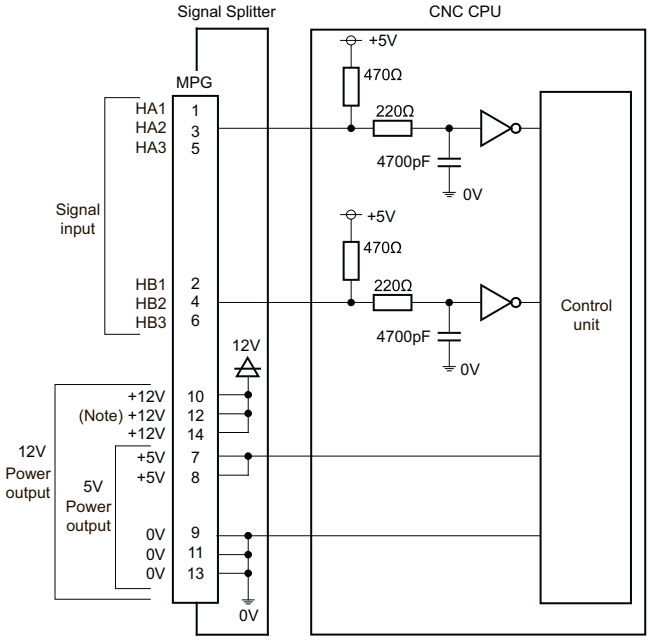
1	IN	HA1
2	IN	HB1
3	IN	HA2
4	IN	HB2
5	IN	HA3
6	IN	HB3
7	OUT	+5VDC
8	OUT	+5VDC

9	OUT	SG(0V)
10	OUT	+12VDC
11	OUT	SG(0V)
12	OUT	+12VDC
13	OUT	SG(0V)
14	OUT	+12VDC
15		---

	5V manual pulse generator (UFO-01-2Z9) input conditions	12V manual pulse generator (HD60) input conditions
Input pulse signal type	HA1 and HB1 phases (with phase difference 90°) (Refer to the waveform below.)	
Input signal voltage	H level 3.5V to 5.25V L level 0V to 0.5V	
Max. input pulse frequency	100kHz	
Pulse generators power supply voltage	5VDC ± 10%	5VDC ± 10%
Current consumption	100mA or less	
Number of pulses per rotation	100 pulse/rev	25 pulse/rev



a.b.c.d.e: HA1 or HB1 rising edge (falling edge) phase difference = T/4 ± T/10
T: HA1 or HB1 cycle (Min. 10μs)



(Note) 12V power is separately required to connect 12V manual pulse generator.
(Refer to 4.9 Connecting the Manual Pulse Generator)

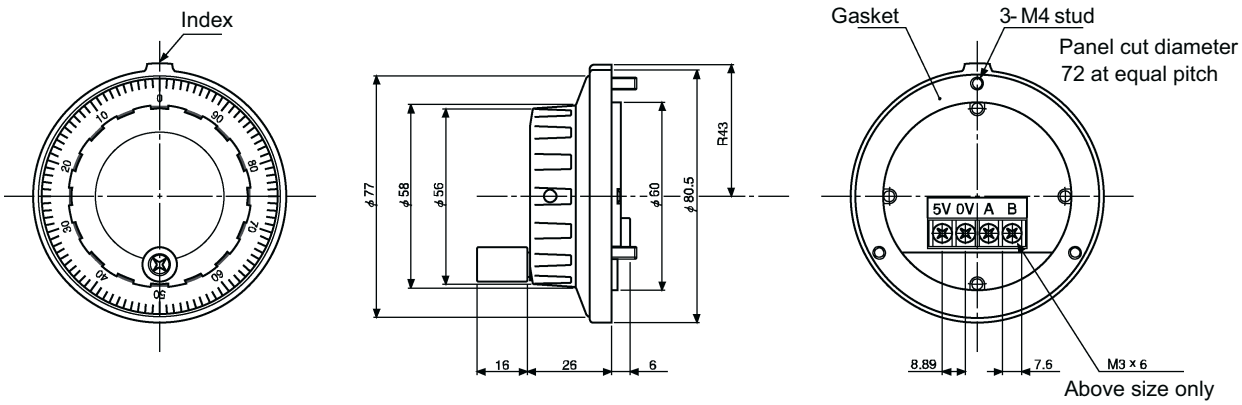
(6) TERMINAL : (Not used)

2.9 Manual Pulse Generator

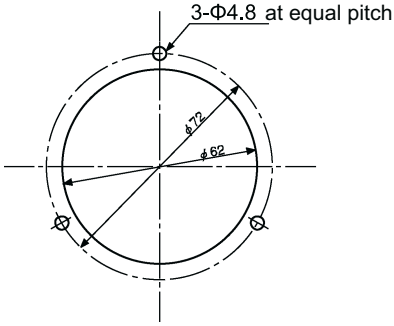
[UFO-01-2Z9]

5V manual pulse generator (100 pulse/rev)

<Outline dimension>



<Panel cut drawing>



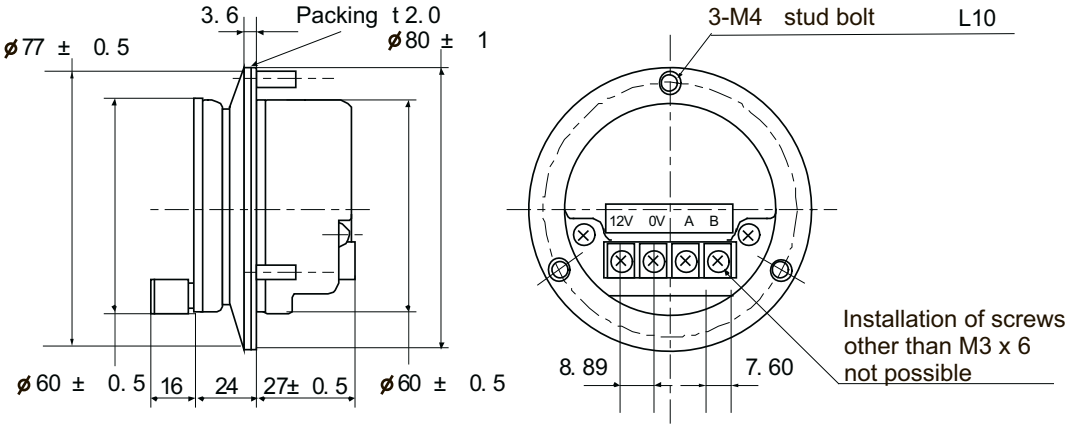
Produced by NIDEC NEMICON CORPORATION

(Note) This product does not comply with MITSUBISHI CNC standard specifications.

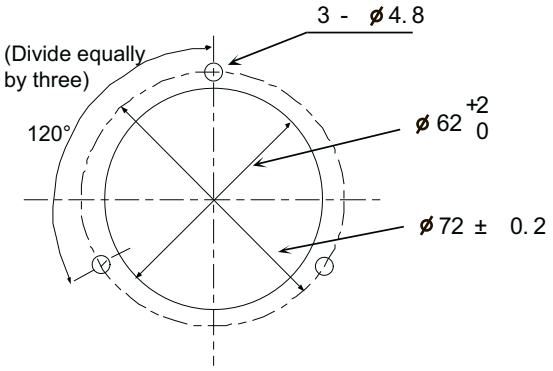
[HD60]

12V manual pulse generator (100 pulse/rev)

<Outline dimension>



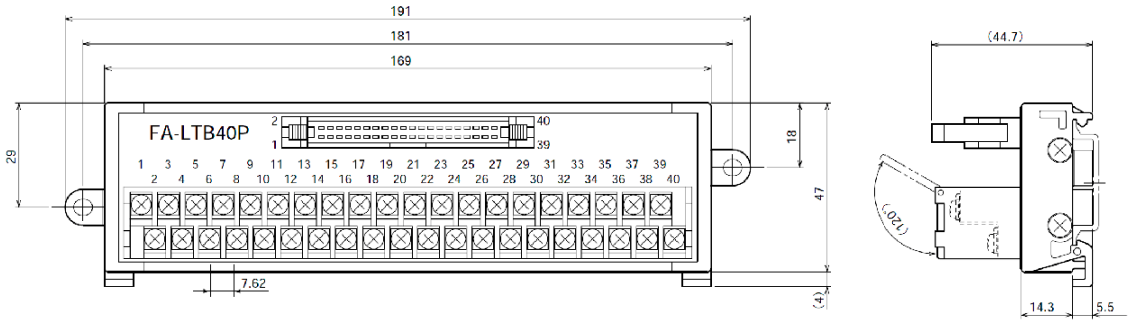
<Panel cut dimension drawing>



2.10 Terminal block for Dual Signal Module (Recommended)

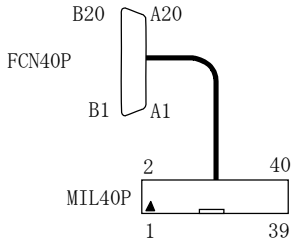
Terminal block converter module FA-LTB40P, produced by MITSUBISHI ELECTRIC ENGINEERING, is recommended to connect the dual signals to the dual signal module. Use the connection cable FA-CBL ☐☐ FMV-M produced by MITSUBISHI ELECTRIC ENGINEERING.

A dual signal module requires two units of terminal converter modules and two cables.

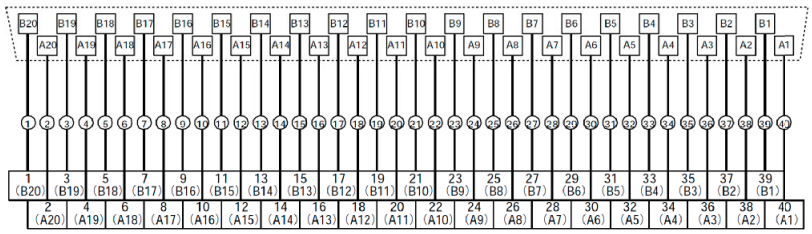


FA-CBL ☐☐ FMV-M cable (length: 05 as 0.5m, 10 as 1m, 20 as 2m, 30 as 3m and 50 as 5m)

Connector and the terminal block



Connection diagram



(Note 1) Connect 24VDC to the terminals No.37 and 39, OV to the terminals No.38 and 40.

(Note2) Input/output cables must be protected against damage and mechanical stress/movement.
The installation must be that short circuits between cores (of multicore cables) cannot be possible or do not lead to hazardous situation.

(Note 3) EMG-Switches must employ 2 NC contacts and be of direct opening type. (IEC60947-5-1 Annex K, IEC60947-5-5)

2.11 I/O Extension Connector Unit

General specifications of I/O Extension connector unit is same as that of GOT. Refer to the instruction manual of GOT you are using.

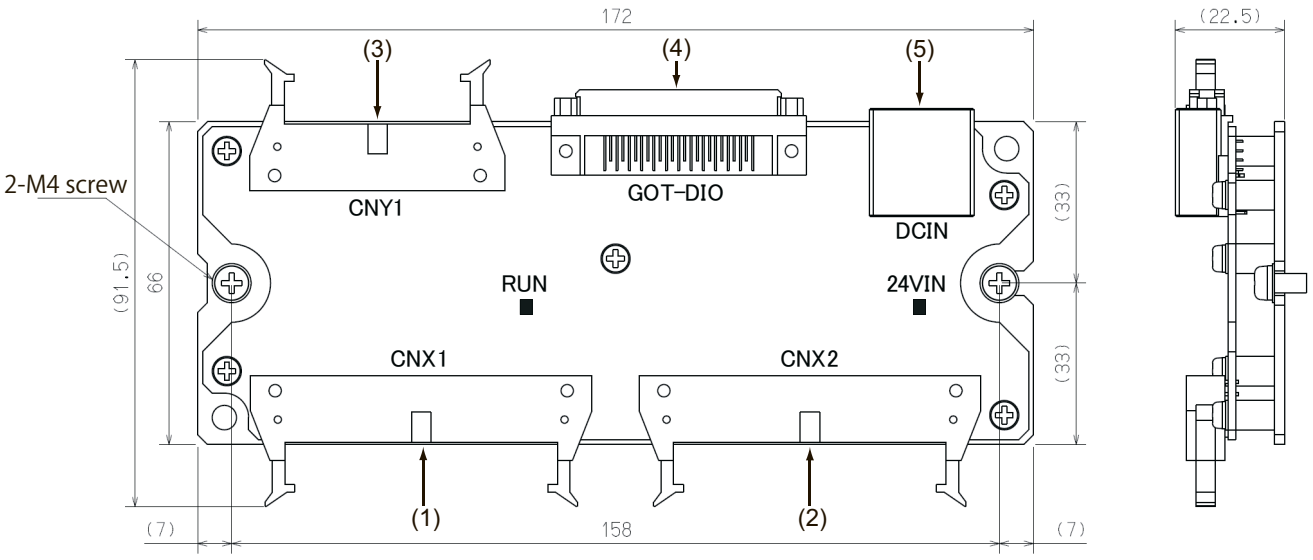
As for input/output specifications, they are basically same as GT15-DIOR unit apart from the number of input points is extended to 64points. Refer to the instruction manual for GT15-DIOR unit.

(Note) This unit is dedicated to GT15-DIOR (sink iput/source output). It cannot be used for GT15-DIO (source input/sink output)

Specifications list

Item	Specification
External connection method	Input connector: MIL-40 pin connector x 2 (CNX1, CNX2) Output connector: MIL-26 pin connector x 1 (CNY1)
Applicable size of wire	Batch solderless type: AWG28 1.27 pitch flat cable Multicore cable solderless type: AQG24-28 twisted cable
External power supply	[Voltage] 24VDC (20.4 - 28.8V, Ripple ratio: Less than 5%)
	[Current] 1.85A
	[Connector] DCIN connector (Supply from CNX1 or CNX3 connector is available) (Applicable size of electric wire: AWG16 - 20)
Connection cable between GT15 and DIOR	H810 cable (Install FCU7-HN831 unit in the same panel as GOT.)
Input method	Dynamic scan method/sink input
The number of input points	64 points (16 points x 4, 4 points of output for scan are used)
Cycle of dynamic scan	13.3ms
Output method	Direct output/source output
The number of output points	16 points + 1point(RUN)
Protection function	Generic output signal: Overload protection function, Overheat protection function (inside GT15-DIOR) RUN output signal: Overload protection function (inside FCU7-HN831) (Recovers automatically when overload or overheat is resolved.)
LED display	24VINDC, RUN output(RUN)
Outline dimension	172 × 66[91.5] × 22.5 (The figure inside brackets indicates the dimension to the tip of the connector.)

Outline dimension and names of each parts



(1) CNX1

Connector: 3432-6002-LCPL * 3M

(Cable side: 7940-□□00SC/3448-7940)

Pin number	Name of the signal	
	B	A
20	X00	X10
19	X01	X11
18	X02	X12
17	X03	X13
16	X04	X14
15	X05	X15
14	X06	X16
13	X07	X17
12	X08	X18
11	X09	X19
10	X0A	X1A
9	X0B	X1B
8	X0C	X1C
7	X0D	X1D
6	X0E	X1E
5	X0F	X1F
4	COM0	COM1
3	COM0	COM1
2	(24VDC)	(0V)
1	(24VDC)	(0V)

(2) CNX2

Connector: 3432-6002-LCPL * 3M
(Cable side: 7940-□□00SC/3448-7940)

Pin number	Name of the signal	
	B	A
20	X20	X30
19	X21	X31
18	X22	X32
17	X23	X33
16	X24	X34
15	X25	X35
14	X26	X36
13	X27	X37
12	X28	X38
11	X29	X39
10	X2A	X3A
9	X2B	X3B
8	X2C	X3C
7	X2D	X3D
6	X2E	X3E
5	X2F	X3F
4	COM2	COM3
3	COM2	COM3
2	(24VDC)	(0V)
1	(24VDC)	(0V)

(3) CNY1

Connector: 3429-5002-LCPL * 3M
(Cable side: 7926-□□00SC/3448-7926)

Pin number	Name of the signal	
	B	A
13	Y00	Y08
12	Y01	Y09
11	Y02	Y0A
10	Y03	Y0B
9	Y04	Y0C
8	Y05	Y0D
7	Y06	Y0E
6	Y07	Y0F
5	0V	0V
4	0V	0V
3	N.C	N.C
2	RUN	N.C
1	0V	N.C

(4) GOT-DIO

Connector: PCS-E50LMD+ * HONDA TSUSHIN KOGYO
(Cable side: PCS-E50FA)

Pin number	Name of the signal	Pin number	Name of the signal
25	XD0E	50	XD0F
24	XD0C	49	XD0D
23	XD0A	48	XD0B
22	XD08	47	XD09
21	XD06	46	XD07
20	XD04	45	XD05
19	XD02	44	XD03
18	XD00	43	XD01
17	XSCN06	42	XSCN07
16	XSCN04	41	XSCN05
15	XSCN02	40	XSCN03
14	XSCN00	39	XSCN01
13	YD0E	38	YD0F
12	YD0C	37	YD0D
11	YD0A	36	YD0B
10	YD08	35	YD09
9	YD06	34	YD07
8	YD04	33	YD05
7	YD02	32	YD03
6	YD00	31	YD01
5	N.C	30	RUN
4	24VDC	29	0V
3	24VDC	28	0V
2	24VDC	27	0V
1	24VDC	26	0V

(5) DCIN

Connector: 2-178313-5 * Tyco Electronics
(Cable side: 2-178288-3)

Pin number	Name of the signal
3	FG
2	0V
1	24VDC

(Note 1) Xxx or Yxx. in this chapter does not indicate the internal device No.

(Note 2) Connect to common signal which is determined for each input signal since dynamic scan method is applied for the input method. (If the common is connected to 24DVC, it does not operate normally.)
X00 to X0F: COM0 is used as the common
X10 to X1F: COM1 is used as the common
X20 to X2F: COM2 is used as the common
X30 to X3F: COM is used as the common

(Note 3) It is recommended to use DCIN as a connecter for 24VDC input, but it is available to supply from CNX1 or CNX2. In this case, make sure to wire more than 2 pins.

(Note 4) Pressure welding connector for multicore cable is also required for a cable side connecter which connects to CNX1, CNX2 or CNY1 connecter.
UFS-□□B-04* YAMAICHI ELECTRONICS

3. Servo/Spindle Drive System

(1) Power supply regenerative type

MDS-D-V1/V2 Series	MDS-DH-V1/V2 Series
200VAC (50Hz)/200 to 230VAC (60Hz) +10% -15%	380 to 440VAC (50Hz)/380 to 480VAC (60Hz) ±10%
MDS-D-V1 1st axis servo drive unit MDS-D-V2 2nd axis servo drive unit MDS-D-SP 1st Spindle drive unit MDS-D-SP2 2nd Spindle drive unit MDS-D-CV Power supply unit	MDS-DH-V1 1st axis servo drive unit MDS-DH-V2 2nd axis servo drive unit MDS-DH-SP Spindle drive unit MDS-DH-CV Power supply unit

(2) Resistance regenerative type

MDS-D-SVJ3/SPJ3 Series
200VAC (50Hz)/200 to 230VAC (60Hz) +10% -15%
MDS-D-SVJ3 1st axis servo drive unit MDS-D-SPJ3 Spindle drive unit

(3) Multi axis integrated regenerative type

MDS-DM Series
200VAC (50Hz)/200 to 230VAC (60Hz) +10% -15%
MDS-DM-V3 3-axis integrated servo drive unit MDS-DM-SPV2/SPV3 Multi axis integrated drive unit

4. CNC Signals (PLC Interface Signals)

The CNC signal includes the following signals.
Refer to "PLC Interface Manual" for detail.

Bit Type Input Signals (CNC->PLC)

System State

24 hours continuous operation
Dual signal unconfirmed after compare error
Output OFF check not complete
Power shutoff notification
SKIP0 Input signal state
SKIP1 Input signal state
SKIP2 Input signal state
SKIP3 Input signal state
Controller ready completion
Servo ready completion
Door open enable
In spindle synchronization
Spindle rotation speed synchronization completion
Spindle phase synchronization completion
Chuck close confirmation
Battery warning
Battery alarm
NC alarm 1
NC alarm 2 (Servo alarm)
In door interlock
Macro single valid
Power OFF required after parameter change
Edited data in processing
Edited data error
NC data sampling completed
ATS varid
Download in progress
Download completed
Download error
PLC axis position switch 1
PLC axis position switch 2
PLC axis position switch 3
PLC axis position switch 4
PLC axis position switch 5
PLC axis position switch 6
PLC axis position switch 7
PLC axis position switch 8
PLC axis position switch 9
PLC axis position switch 10
PLC axis position switch 11
PLC axis position switch 12
PLC axis position switch 13
PLC axis position switch 14
PLC axis position switch 15
PLC axis position switch 16
APLC input signal 1-32

Axis State

Servo ready
Axis selection
In axis plus motion
In axis minus motion
1st reference position reached
2nd reference position reached
3rd reference position reached
4th reference position reached
Near reference position
NC axis up-to-speed
Zero point initialization set completed
Zero point initialization set error completed
In zero point initialization
Zero point initialization incomplete
In current limit
Current limit reached
Unclamp command
In-position
In multi-step speed monitor
Multi-step speed monitor mode output 1
Multi-step speed monitor mode output 2
Axis switching invalid status
In PLC axis control

Part System State

In jog mode
In handle mode
In incremental mode
In manual arbitrary feed mode
In reference position return mode
In automatic initial set mode
In memory mode
In MDI mode
In automatic operation "run"
In automatic operation "start"
In automatic operation "pause"
In "reset"
In manual arbitrary feed
In rewind
Motion command completion
All axes in-position
All axes smoothing zero
Manual arbitrary feed completion
External search finished
In rapid traverse
In cutting feed
In tapping
In thread cutting
In synchronous feed
In constant surface speed
In skip
In reference position return
F 1-digit commanded
In tool life management
Tool life over
NC alarm 3 (Program error)
NC alarm 4 (Operation error)
Search & start (error)
Search & start (search)
Illegal axis selected
F 1-digit No. code 1
F 1-digit No. code 2
F 1-digit No. code 4
Waiting between part systems
In hypothetical axis command mode
M code independent output M00
M code independent output M01
M code independent output M02
M code independent output M30
M function strobe 1
M function strobe 2
M function strobe 3
M function strobe 4
Manual numerical command
Tool change position return completion
New tool change
T function strobe 1
2nd M function strobe 1
S function strobe 1
S function strobe 2
S function strobe 3
S function strobe 4
S function strobe 5
S function strobe 6
S function strobe 7
Position switch 1
Position switch 2
Position switch 3
Position switch 4
Position switch 5
Position switch 6
Position switch 7
Position switch 8

Waiting for data to be downloaded
Tap retract possible
No. of work machining over
Power shutoff movement over
Position switch 9
Position switch 10
Position switch 11
Position switch 12
Position switch 13
Position switch 14
Position switch 15
Position switch 16

Spindle State

S command gear No. illegal
S command max./min. command value over
S command no gear selected
Spindle speed upper limit over
Spindle speed lower limit over
Spindle gear shift command 1
Spindle gear shift command 2
Current detection
Speed detection
In spindle alarm
Zero speed
Spindle up-to-speed
Spindle in-position
In L coil selection
Spindle ready-ON
Spindle servo-ON
In spindle forward run
In spindle reverse run
Z-phase passed
Position loop in-position
In spindle torque limit
In spindle multi-step speed monitor
In spindle multi-step speed monitor output 1
In spindle multi-step speed monitor output 2

Data Type Input Signals (CNC->PLC)

System State

KEY IN
Speed monitor door open possible
1st handle pulse counter
2nd handle pulse counter
3rd handle pulse counter
CRT display information
Emergency stop cause
User macro output #1132 (Controller -> PLC)
User macro output #1133 (Controller -> PLC)
User macro output #1134 (Controller -> PLC)
User macro output #1135 (Controller -> PLC)
CNC software version code
Battery drop cause
Temperature warning cause
Spindle synchronization phase error 1
Spindle synchronization phase error 2
Spindle synchronization phase error output
Spindle synchronization Phase error monitor
Spindle synchronization Phase error monitor (lower limit)
Spindle synchronization Phase error monitor (upper limit)
Spindle synchronization Phase offset data
APLC input signal 1-10
NC exclusive instruction (DDWR/DDRD) error ZR device No.
GOT window data changeover completion

Part System State

External search status
M code data 1
M code data 2
M code data 3
M code data 4
S code data 1
S code data 2
S code data 3
S code data 4
T code data 1
2nd M function data 1
Tool No.
Group in tool life management No. of work machining(current value)
Near reference position (per reference position)
Tool life usage data
No. of work machining(maximum value)
Error code output
Error code output extension
S code data 5
S code data 6
S code data 7
User Macro output #1132 (Controller -> PLC)
User Macro output #1133 (Controller -> PLC)
User Macro output #1134 (Controller -> PLC)
User Macro output #1135 (Controller -> PLC)
Chopping status
Chopping error No.
Chopping axis

Axis State

Thermal expansion compensation amount
Servo motor temperature

Spindle State

Spindle command rotation speed input
Spindle command final data (Rotation speed)
Spindle command final data (12-bit binary)
Spindle actual speed
Spindle motor temperature

Bit Type Output Signals (PLC->CNC)

System Command

Contactor shutoff test signal
Dual signals check start
Output OFF check
Integration time input 1
Integration time input 2
Data protect key 1
Data protect key 2
Data protect key 3
CRT changeover completion
Display changeover $1
Display changeover $2
NC data sampling trigger
Saving operation history data
Edited data recovery confirmation
PLC emergency stop
Door open I
Door open II
PLC axis control buffering mode valid
PLC axis 1st handle valid
PLC axis 2st handle valid
PLC axis 3st handle valid
Spindle synchronization cancel
Chuck close
Spindle synchronization
Spindle phase synchronization
Spindle synchronous rotation direction
Phase shift calculation request
Phase offset request

Error temporary cancel
PLC axis near point detection 1st axis
PLC axis near point detection 2nd axis
PLC axis near point detection 3rd axis
PLC axis near point detection 4th axis
PLC axis near point detection 5th axis
PLC axis near point detection 6th axis
PLC axis near point detection 7th axis
PLC axis near point detection 8th axis
PLC axis control valid 1st axis
PLC axis control valid 2nd axis
PLC axis control valid 3rd axis
PLC axis control valid 4th axis
PLC axis control valid 5th axis
PLC axis control valid 6th axis
PLC axis control valid 7th axis
PLC axis control valid 8th axis
Download request
APLC output signal 1-32

Axis Command

Control axis detach
Servo OFF
Mirror image
External deceleration +
External deceleration -
Automatic interlock +
Automatic interlock -
Manual interlock +
Manual interlock -
Automatic machine lock
Manual machine lock
Feed axis selection +
Feed axis selection -
Manual/Automatic simultaneous valid
Control axis detach 2
Current limit changeover
Droop release request
Zero point initialization set mode
Zero point initialization set start
Unclamp completion
Multi-step speed monitor request
Multi-step speed monitor mode input 1
Multi-step speed monitor mode input 2
Counter zero
PLC axis switching

Part System Command

Jog mode
Handle mode
Incremental mode
Manual arbitrary feed mode
Reference position return mode
Automatic initialization mode
Program operation mode (Memory mode)
FTP mode
EDIT mode
MDI mode
Automatic operation "start" command (Cycle start)
Automatic operation "pause" command (Feed hold)
Single block
Block start interlock
Cutting block start interlock
Dry run
Error detect
NC reset 1
NC reset 2
Reset & rewind
Chamfering
Automatic restart
External search strobe
M function finish 1
M function finish 2
Tool length measurement 1
Tool length measurement 2 (L system)
Synchronization correction mode
Macro interrupt
Rapid traverse
Manual absolute
Recalculation request
Program display during operation
Optional block skip 1
Reference position selection code 1
Reference position selection code 2
Reference position selection method
Optional block skip 2
Optional block skip 3
Optional block skip 4
Optional block skip 5
Optional block skip 6
Optional block skip 7
Optional block skip 8
Optional block skip 9
1st handle axis selection code 1
1st handle axis selection code 2
1st handle axis selection code 4
1st handle axis selection code 8
1st handle axis selection code 16
1st handle valid
2nd handle axis selection code 1
2nd handle axis selection code 2
2nd handle axis selection code 4
2nd handle axis selection code 8
2nd handle axis selection code 16
2nd handle valid
3rd handle axis selection code 1
3rd handle axis selection code 2
3rd handle axis selection code 4
3rd handle axis selection code 8
3rd handle axis selection code 16
3rd handle valid
Override cancel
Manual override method selection
Miscellaneous function lock
Tap retract
Reference position retract
Cutting feedrate override code 1
Cutting feedrate override code 2
Cutting feedrate override code 4
Cutting feedrate override code 8
Cutting feedrate override code 16
2nd cutting feedrate override valid
Cutting feedrate override method selection
Rapid traverse override code 1
Rapid traverse override code 2
Rapid traverse override method selection
Manual feedrate code 1
Manual feedrate code 2
Manual feedrate code 4
Manual feedrate code 8
Manual feedrate code 16
Manual feedrate method selection
Feedrate least increment code 1
Feedrate least increment code 2
Jog synchronous feed valid
Jog handle synchronous
Current limit mode 1
Current limit mode 2
Handle/incremental feed multiplication code 1
Handle/incremental feed multiplication code 2
Magnification valid for each handle
Handle/incremental feed multiplication code 4
Handle/incremental feed magnification method selection
Tool alarm 1 /Tool skip 1
Tool alarm 2
Usage data count valid
Tool life management input
Tool change reset
Manual arbitrary feed 1st axis

selection code 1
Manual arbitrary feed 1st axis selection code 2
Manual arbitrary feed 1st axis selection code 4
Manual arbitrary feed 1st axis selection code 8
Manual arbitrary feed 1st axis selection code 16
Manual arbitrary feed 1st axis valid
Manual arbitrary feed 2nd axis selection code 1
Manual arbitrary feed 2nd axis selection code 2
Manual arbitrary feed 2nd axis selection code 4
Manual arbitrary feed 2nd axis selection code 8
Manual arbitrary feed 2nd axis selection code 16
Manual arbitrary feed 2nd axis valid
Manual arbitrary feed 3rd axis selection code 1
Manual arbitrary feed 3rd axis selection code 2
Manual arbitrary feed 3rd axis selection code 4
Manual arbitrary feed 3rd axis selection code 8
Manual arbitrary feed 3rd axis selection code 16
Manual arbitrary feed 3rd axis valid
Manual arbitrary feed smoothing off
Manual arbitrary feed axis independent
Manual arbitrary feed EX.F/MODAL.F
Manual arbitrary feed G0/G1
Manual arbitrary feed MC/WK
Manual arbitrary feed ABS/INC
Manual arbitrary feed stop
Manual arbitrary feed strobe
2nd reference position return interlock
Search & start
Inclined axis control: no z axis compensation
Hypothetical axis command mode
Chopping
Chopping parameter valid
Compensation method selection
Operation mode selection
Rapid traverse override valid

Spindle Command

Gear shift completion
Spindle override code 1
Spindle override code 2
Spindle override code 4
Spindle override method selection
Spindle gear selection code 1
Spindle gear selection code 2
Spindle stop
Spindle gear shift
Spindle orientation
Spindle forward run start
Spindle reverse run start
Spindle forward run index
Spindle reverse run index
Spindle orientation command
L coil selection
Spindle torque limit 1
Spindle torque limit 2
Spindle torque limit 3
Spindle multi-step monitor request
Spindle multi-step speed monitor mode input 1
Spindle multi-step speed monitor mode input 2
External axis speed clamp

Data Type Output Signals (PLC->CNC)

System Command

Speed monitor mode
PLC axis droop release invalid axis
KEY OUT
Speed monitor mode
User macro input #1032 (PLC -> Controller)
User macro input #1033 (PLC -> Controller)
User macro input #1034 (PLC -> Controller)
User macro input #1035 (PLC -> Controller)
PLC version code
1st axis index
2nd axis index
3rd axis index
4th axis index
5th axis index
6th axis index
7th axis index
8th axis index
9th axis index
10th axis index
11th axis index
12th axis index
13th axis index
14th axis index
15th axis index
16th axis index
Spindle synchronization Basic spindle selection
Spindle synchronization
Synchronous spindle selection
Spindle synchronization Phase shift amount
PLC version code (method 2)
APLC output data 1-10
GOT window Data changeover request

Part System Command

1st cutting feedrate override
2nd cutting feedrate override
Rapid traverse override
Manual feedrate
1st handle/incremental feed magnification
2nd handle feed magnification
3rd handle feed magnification
Manual arbitrary feed 1st axis travel amount
Manual arbitrary feed 2nd axis travel amount
Manual arbitrary feed 3rd axis travel amount
OT ignored
Near-point dog ignored
Tool group No. designation
Synchronization control operation method
Droop release invalid axis
Search & start program No.
Each axis reference position selection
Workpiece coordinate offset measurement compensation No.
Selected tool No.
External search device No.
External search program No.
External search sequence No.
External search block No.
User Macro input #1032 (PLC -> Controller)
User Macro input #1033 (PLC -> Controller)
User Macro input #1034 (PLC -> Controller)
User Macro input #1035 (PLC -> Controller)
Chopping override
Chopping axis selection
Upper dead point designation (L)
Upper dead point designation (H)

Lower dead point designation (L)
Lower dead point designation (H)
Number of cycles designation
Data No.

Axis Command

External machine coordinate system compensation data
Thermal expansion offset compensation amount
Thermal expansion max. compensation amount
External deceleration speed selection

Spindle Command

Spindle command rotation speed output
S command override
Multi-point orientation position data

Classified Under Purpose
(CNC->PLC)
(PLC->CNC)

PLC axis state
PLC axis control
Window result information
Window command
Data registered to magazine for M system
Tool life management (M system)
Safety observing
PLC constants
PLC bit selection
PLC axis indexing interface
Special relay/register signals

II FUNCTIONAL SPECIFICATIONS

1. Control Axes

The NC axis, spindle, PLC axis are generically called the control axis.
The NC axis can be manually or automatically operated using the machining program.
The PLC axis can be controlled using the sequence program.

1.1 Control Axes

1.1.1 Number of Basic Control Axes (NC axes)

M system : 3 axes **L system : 2 axes**

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

M system : 16 axes **L system : 16 axes**

A number of axes that are within the maximum number of control axes, and that does not exceed the maximum number given for the NC axis, spindle and PLC axis can be used.
For example, if 16 NC axes are used, this alone is the maximum number of control axes, so a spindle and PLC axis cannot be connected.

1.1.2.1 Max. number of NC axes (in total for all the part systems)

M system : 16 axes **L system : 16 axes**

1.1.2.2 Max. number of spindles

M system : 7 axes **L system : 4 axes**

1.1.2.3 Max. number of PLC axes

M system : 8 axes **L system : 8 axes**

1.1.4 Max. number of PLC indexing axes

M system : 8 axes **L system : 8 axes**

1.1.5 Number of Simultaneous Contouring Control Axes

Simultaneous control of up to four axes or less is possible in the same part system.
However, for actual use, the machine tool builder specification will apply.

M system : 4 axes **L system : 4 axes**

1.1.6 Max. Number of NC Axes in a Part System

M system : 8 axes **L system : 8 axes**

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

1.2 Control Part System

1.2.1 Standard Number of Part Systems

M system : 1 part system L system : 1 part system

The standard number of part systems is one.

1.2.2 Max. Number of Part Systems

M system : Δ7 part systems L system : Δ3 part systems

The maximum number of part systems for lathe system is three, and for machining center is seven.
For actual use, the machine tool builder specification will apply.

1.3 Control Axes and Operation Modes

1.3.2 Memory Mode

M system : O L system : O

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

1.3.3 MDI Mode

M system : O L system : O

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 performing screen operations.

1.3.102 High-speed program server mode

M system : Δ L system : Δ

This function allows a high-speed transfer and operation of machining programs to the large capacity buffer memory in a CNC CPU, using the Ethernet FTP function. And the operation requires an FTP server, such as PC, or an Ethernet connection with GTO (with FTP server function) connected with a CF card.

2. Input Command

2.1 Data Increment

2.1.1 Least command increment

2.1.1.1 Least command increment: 1 μm

M system : ○ **L system : ○**

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 μm

M system : Δ **L system : Δ**

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

The data increment handled in the controller include the least input increment, least command increment and least detection increment. Each type is set with parameters.

(1) The least input increment indicates the increment handled in the internal processing of the controller. The counter and tool offset data, etc., input from the screen is handled with this increment. This increment is applied per part system (all part systems, PLC axis).

Increment type	Input increment (parameter)	Metric unit system		Inch unit system	
		Linear axis (Unit = mm)	Rotary axis (Unit = °)	Linear axis (Unit = inch)	Rotary axis (Unit = °)
Least input increment	B	0.001	0.001	0.0001	0.001
	C	0.0001	0.0001	0.00001	0.0001

(Note 1) 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.

Increment type	Command increment (parameter)	Metric unit system		Inch unit system	
		Linear axis (Unit = mm)	Rotary axis (Unit = °)	Linear axis (Unit = inch)	Rotary axis (Unit = °)
Command increment	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 1) The inch and metric systems cannot be used together.

(3) The least detection increment indicates the detection increment of the NC axis and PLC axis detectors. The increment is determined by the detector being used.

2.2 Unit System

2.2.1 Inch/Metric Changeover

M system : Δ **L system : Δ**

The unit systems of the data handled in the controller include the metric system and inch system. The type can be designated with the parameters and machining program. The unit system can be set independently for the (1) Program command, (2) Setting data such as offset amount and (3) Parameters.

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

(Note 1) For the angle data, 1.0 means 1 degree (°) regardless of the unit system.

Parameter \ Data		Machining program		Screen data (Offset amount, etc.)	Parameter
I_inch	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		
M_inch	0	Not affected		Not affected	Metric unit system
	1				Inch unit system

(Note 1) The parameter changeover is valid after the power is turned ON again.
(Note 2) Even if parameter "I_inch" is changed, the screen data (offset amount, etc.) will not be automatically converted.
(Note 3) When the power is turned ON or resetting is performed, the status of the G20/G21 modal depends on the "I_G20" parameter setting.

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 specification manual explains the G function with G-code series 3 as standard.

2.3.1.1 Format 1 for Lathe

M system : - **L system : ○**

2.3.1.2 Format 2 for Lathe

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 : O **L system : O**

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

(1) Decimal point input type I (When parameter #1078 Decpt2 is 0.)
When axis coordinates and other data are supplied in machining program commands, the assignment of the program data can be simplified by using the decimal point input. The minimum digit of a command not using a decimal point is the same as the least command increment.
Usable addresses can be applied not only to axis coordinate values 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 (When parameter #1078 Decpt2 is 1.)
As opposed to type I, when there is no decimal point, the final digit 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 (Note 1)	Y200μm, F100mm/min	Y200mm, F100mm/min
G4	X1.5	Dwell 1.5 s	←
G4	X2	2ms	2s

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

2.4.2 Absolute/Incremental Command

M system : O **L system : O**

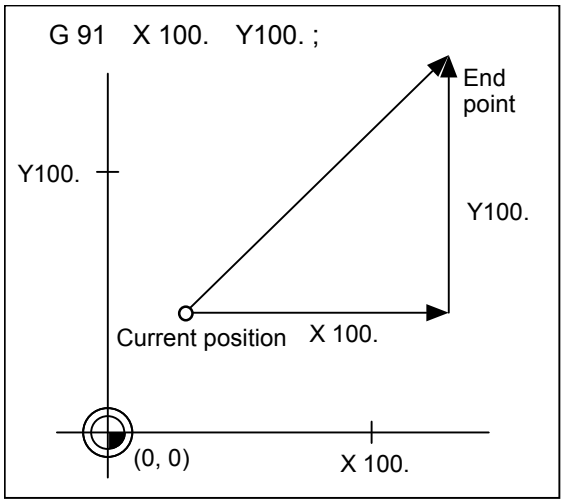
(1) M system
When axis coordinate data is issued in a machining program command, either the incremental command method (G91) that commands a relative distance from the current position or the absolute command method (G90) that moves to a designated position in a predetermined coordinate system can be 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 value command)
G91 ... Incremental command (incremental value command)
These G codes can be commanded multiple times in one block.

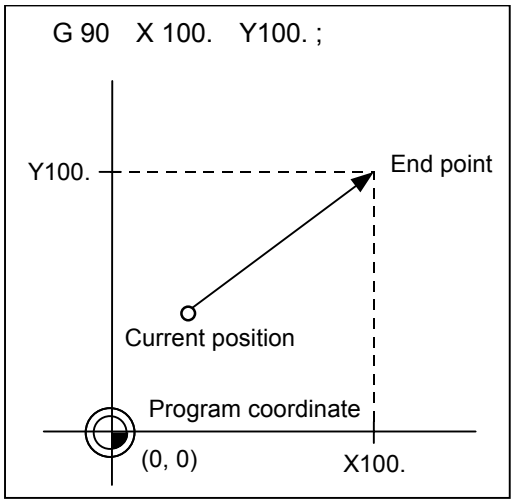
Example

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

(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 value command) (Absolute value command)





(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.

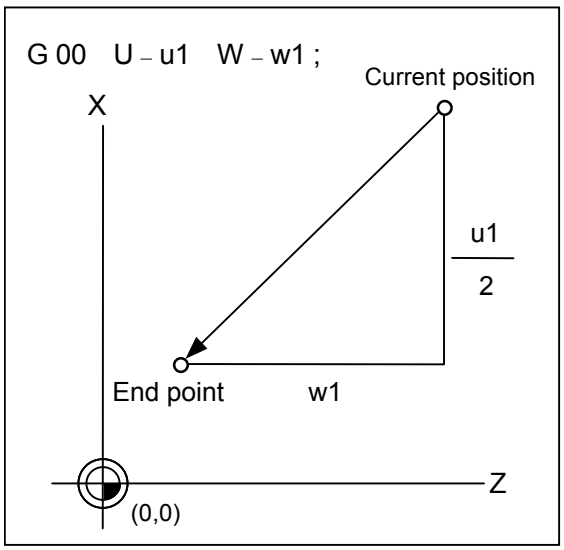
When issuing an incremental value command, the axis address to be commanded as the incremental axis name is registered in the parameter. However, the arc radius designation (R) and arc center designation (I, J, K) always use incremental designations.

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

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

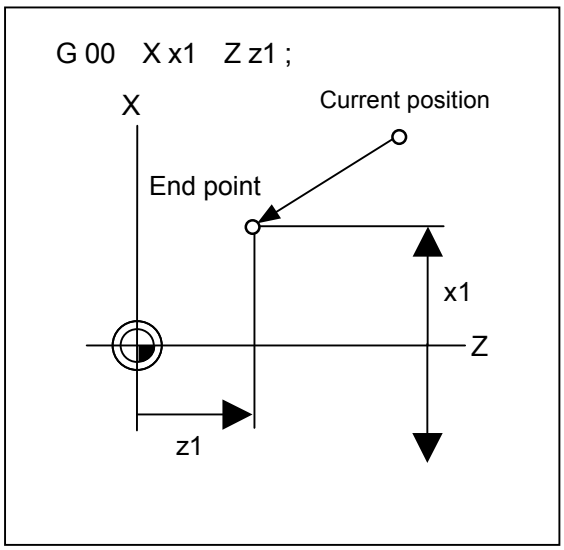
Example	G00	X100.	W200.	;
		Absolute value	Incremental value	

(Incremental value command)



The above drawing shows the case for the diameter command.

(Absolute value command)



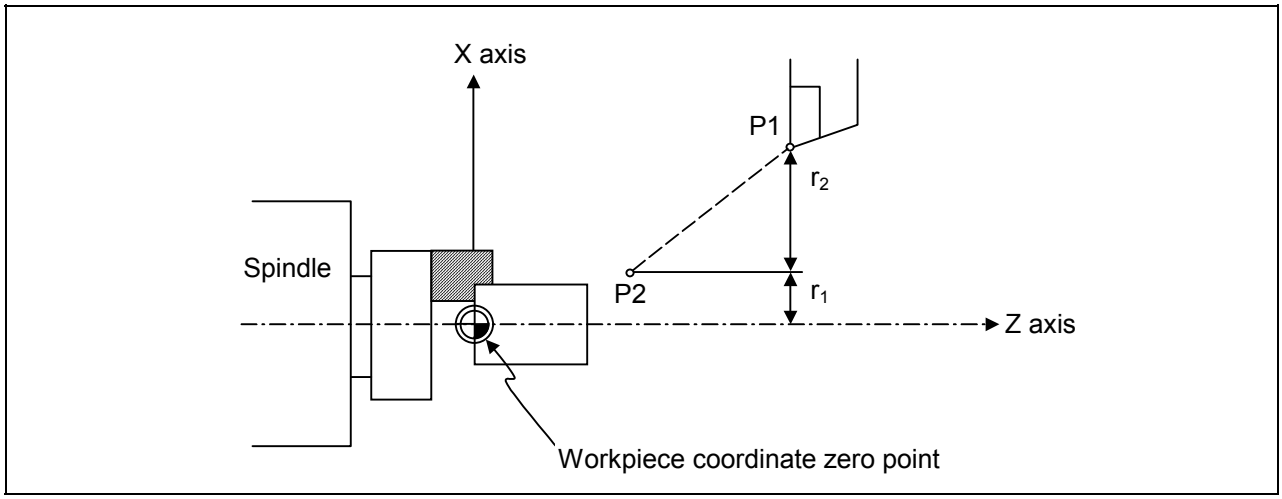
The above drawing shows the case for the diameter command.

(Note 1) In addition to the above command method using the above axis addresses, the absolute value command and incremental value command can be switched by commanding the G code (G90/G91). (Select with the parameters.)

2.4.3 Diameter/Radius Designation

M system : - **L system : O**

For the axis command value, the radius designation or diameter designation can be changed with parameters.
When the diameter designation is selected, the scale of the length of the selected axis is doubled. (Only 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

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.

Radius and diameter commands

3. Positioning/Interpolation

3.1 Positioning

3.1.1 Positioning

M system : O **L system : O**

This function carries out positioning at high speed using a rapid traverse rate with the movement command value given in the program.

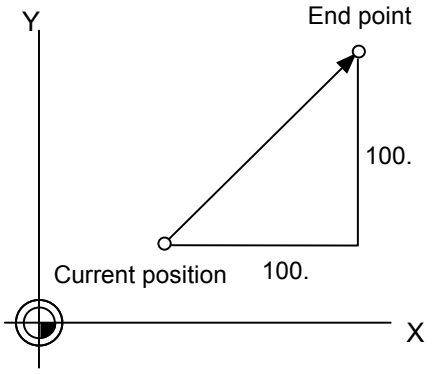
G00 Xx1 Yy1 Zz1 ; (Also possible for additional axes A, B, C, U, V, W simultaneously) x1, y1, z1: numerical values denoting the position data

The above command positions the tool by rapid traverse. 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 for each axis can be set independently with parameters.
(2) The number of axes which can be driven 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)

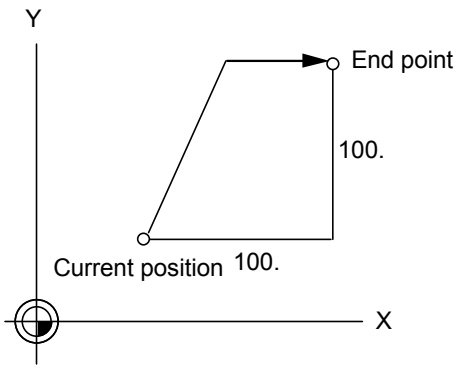
(Example) Linear type (Moves lineary to the end point.)

G 00 G 91 X 100. Y 100. ;



(Example) Non-linear type (Each axis moves at each parameter speed.)

G 00 G 91 X 100. Y 100. ;



(Note 1) If the acceleration/deceleration conditions differ between the axes, the path will not be linear to the end point even when using the linear type.

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

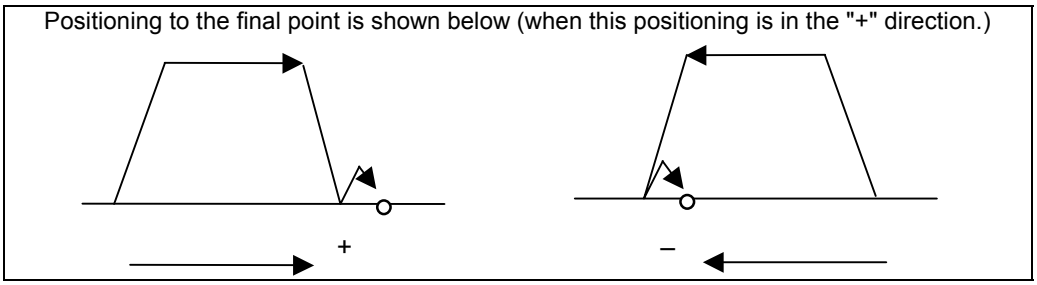
3.1.2 Unidirectional Positioning

M system : Δ **L system : -**

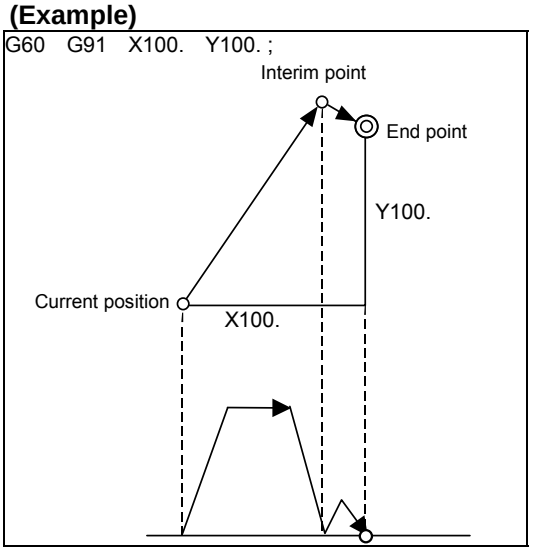
The G60 command always moves the tool to the final position in the direction determined by parameters. The tool can be positioned without backlash.

G60 Xx1 Yy1 Zz1 ; (Also possible for additional axes A, B, C, U, V, W simultaneously) x1, y1, z1: numerical values denoting the position data

With the above command, the tool is first moved to a position distanced from the end point position by an amount equivalent to the creep distance (parameter setting) 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.



(Example)



(1) The rapid traverse rate for each axis is the value set with parameters as the G00 speed.
(2) The vector speed to the interim point is the value produced by combining the distance and respective speeds.
(3) The creep distance of the distance between the interim and end points can be set independently for each axis by "parameters".

(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 : O **L system : O**

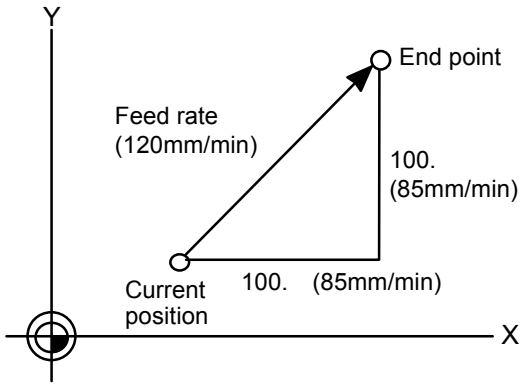
Linear interpolation is a function that moves a tool linearly by the movement command value supplied in the program at the cutting feed rate designated by the F code.

G01 Xx1 Yy1 Zz1 Ff1 ; (Also possible for additional axes A, B, C, U, V, W simultaneously)
x1, y1, z1 : numerical values denoting the position data
f1 : numerical value denoting the 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 ;



(1) The cutting feed rate command moves the tool in the vector direction.
(2) The component speeds of each axis are determined by the proportion of respective command values to the actual movement distance with linear interpolation.

(1) The number of axes which can be driven 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)

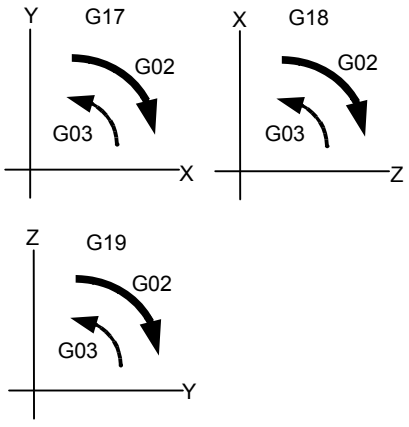
M system : O **L system : O**

(1) Circular interpolation with I, J, K commands
This function moves a tool along a circular arc on the plane with movement command value supplied in the program.

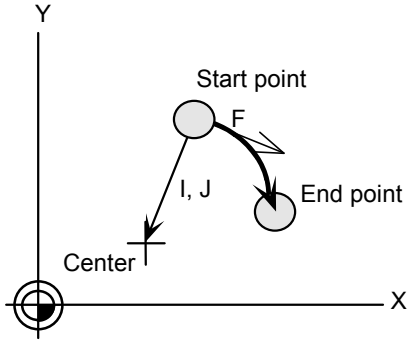
G02(G03) Xx1 Yy1 Ii1 Jj1 Ff1 ; (Also possible for additional axes A, B, C, U, V, W)	
G02, G03	: Arc rotation direction
Xx1, Yy1	: End point coordinate values
Ii1, Jj1	: Arc center coordinate values
Ff1	: Feed rate

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) 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".

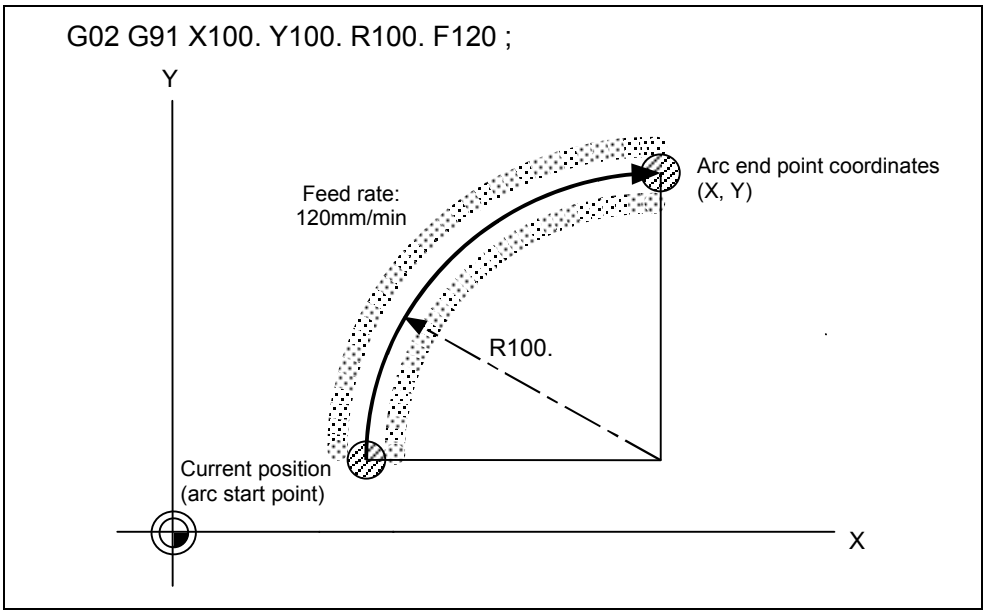
(2) R-specified circular interpolation
Besides the designation of the arc center coordinates using the above-mentioned I, J and K commands, arc commands can also be issued by designating the arc radius directly.

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 values 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)



(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 : Δ **L system : Δ**

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	: Pitch No.
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 coordinate 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.

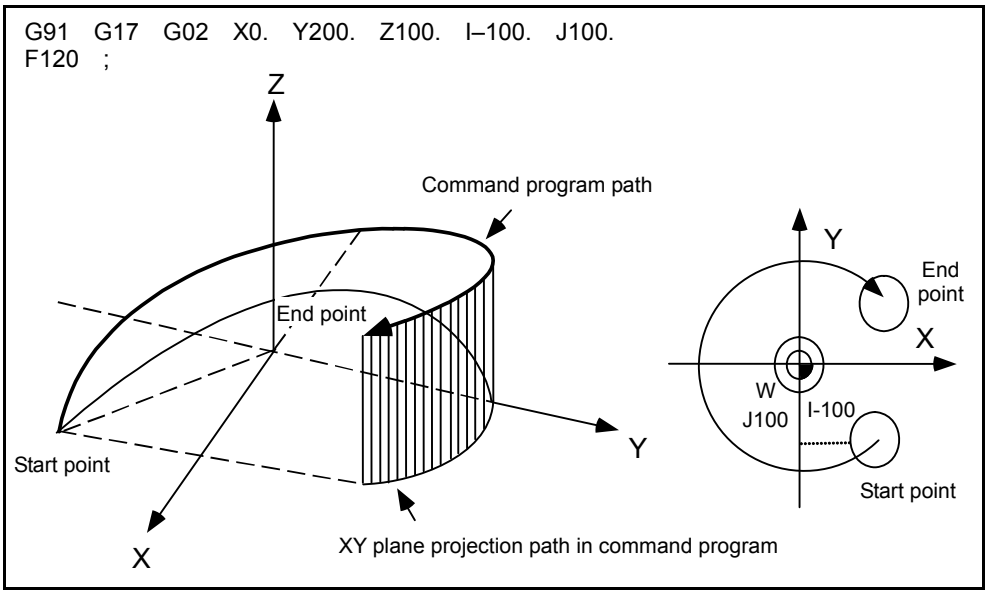
$I1 = z1/((2\pi \bullet p1 + \theta)/2\pi)$

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

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

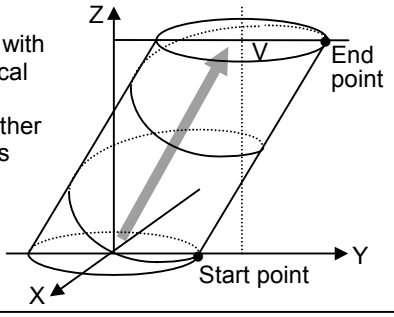
The combination of the axes which can be commanded simultaneously depends on the specifications. The axes can be used in any combination under the specifications. The feed rate is controlled so that the tool always moves at a speed along the circumference of the circle.

(Example)



(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 result 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.



3.2.101 Hypothetical Linear Axis Control

M system : Δ **L system : -**

Using 1 linear axis and 1 rotary axis, this function composes a hypothetical linear axis which orthogonally intersects the real linear axis. Thus, 3-dimensional positioning and compensation operation can be executed by configuring two linear axes and one rotation axis.
This function reduces actual linear axes, so movement mechanism will be eliminated and the machine configuration can be simplified. In the figure below, the movement mechanism of X axis direction which intersects with Y axis is not required. Although a hypothetical axis has no actual linear axis, it needs setting as CNC control axis. There are three actual axes and two hypothetical axes, so total of five CNC control axes are required for the figure below. A mode to control a hypothetical axis is called hypothetical axis command mode and a mode to control actual axis is called actual axis command mode. Each mode can be switched with a control signal from PLC I/F.
This function is valid both automatic operation and manual feed.

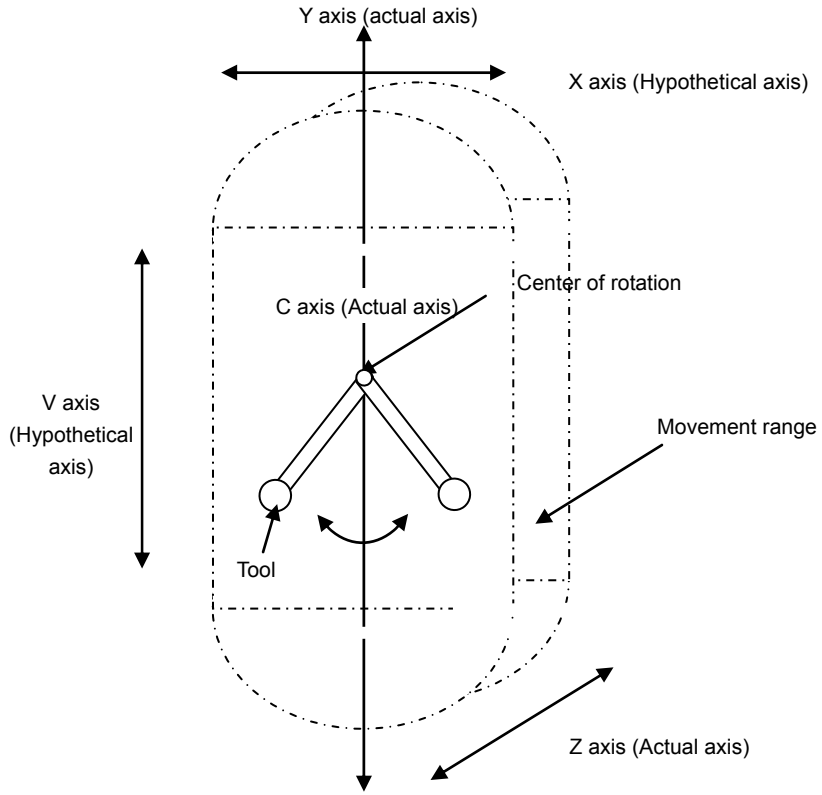
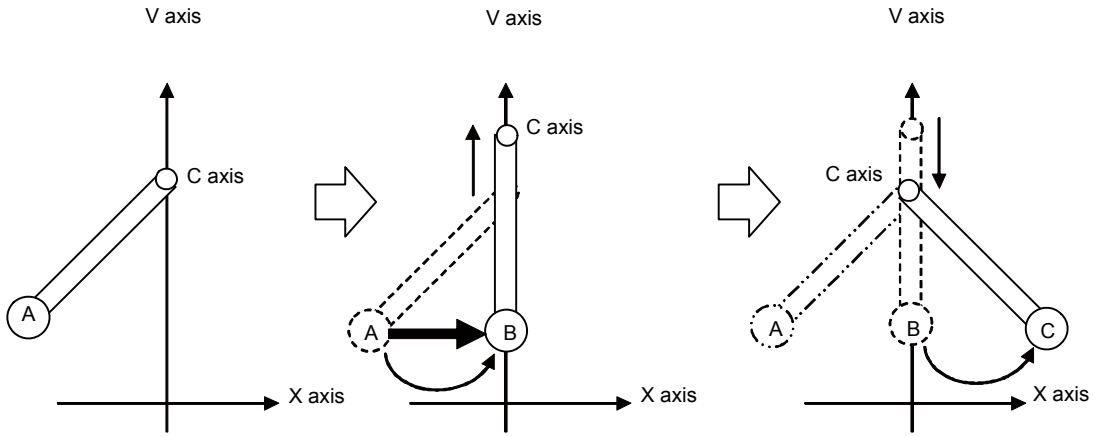


Image of Hypothetical linear axis

Movement of hypothetical X axis is realized by controlling the rotation axis (C axis) and the linear axis (V axis).
(1) V axis moves up with rotating C axis in counterclockwise. (Move from A to B)
(2) After that, V axis moves down with rotating C axis in counterclockwise. (Move from B to C)

This enables the X axis to operate as if it has moved to right.



Operation example of hypothetical X axis which is controlled by the rotation axis C and the linear axis V

Current position
Previous position
Last but one position

4. 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 using 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 input increment	B	C
Metric input	1~1000000 (mm/min, °/min)	1~100000 (mm/min, °/min)
Inch input	1~39370 (inch/min)	1~3937 (inch/min)

Least input increment B : 0.001 mm (0.0001 inch)
Least input increment C : 0.0001 mm (0.00001 inch)

[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 input increment	B	C
Metric input	1~1000000 (mm/min, °/min)	1~100000 (mm/min, °/min)
Inch input	1~39370 (inch/min)	1~3937 (inch/min)

Least input increment B : 0.001 mm (0.0001 inch)
Least input increment C : 0.0001 mm (0.00001 inch)

4.1.2 Cutting Feed Rate (m/min)

M system : 1000 **L system : 1000**

[M system]

This function specifies the feedrate of the cutting commands, and gives a command for a feed amount per spindle rotation or feed amount per minute.
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 input increment	B	C
Metric input	1~1000000 (mm/min, °/min)	1~100000 (mm/min, °/min)
Inch input	1~39370 (inch/min)	1~3937 (inch/min)

Least input increment B : 0.001 mm (0.0001 inch)
Least input increment C : 0.0001 mm (0.00001 inch)

- The cutting feed rate is effective for G01, G02, G03, G33 commands, etc. As to others, 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 input increment	B	C
Metric input	1~1000000 (mm/min, °/min)	1~100000 (mm/min, °/min)
Inch input	1~39370 (inch/min)	1~3937 (inch/min)

Least input increment B : 0.001 mm (0.0001 inch)
Least input increment C : 0.0001 mm (0.00001 inch)

- The cutting feed rate is effective for G01, G02, G03, G33 commands, etc. As to others, refer to 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 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 using external signals.
The manual feed rate signals from the PLC includes two methods, the code method and numerical value 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 : O **L system : O**

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

M system : O **L system : O**

[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)

Least input increment		(B) 0.001 mm	(C) 0.0001 mm
F command increment (mm/min)	**without decimal point** **with decimal point**	F1 = 1 mm/min F1. = 1 mm/min	F1 = 1 mm/min F1. = 1 mm/min
Command range (mm/min)		0.01~1000000.000	0.001~100000.000

Inch input (inch)

Least input increment		(B) 0.0001 inch	(C) 0.00001 inch
F command increment (inch/min)	**without decimal point** **with decimal point**	F1 = 1 inch/min F1. = 1 inch/min	F1 = 1 inch/min F1. = 1 inch/min
Command range (inch/min)		0.001~100000.0000	0.001~10000.0000

- When commands without a decimal point have been assigned, it is not possible to assign commands under 1 mm/min (or 1 inch/min). To assign commands under 1 mm/min (or 1 inch/min), ensure that commands are assigned with a decimal point.
- The initial status after power-ON can be set to asynchronous feed (per-minute-feed) by setting the "Initial synchronous feed" parameter to OFF.
- The F command increments are common to all part systems.

[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)

Least input increment		(B) 0.001 mm	(C) 0.0001 mm
F command increment (mm/min)	**without decimal point** **with decimal point**	F1 = 1 mm/min F1. = 1 mm/min	F1 = 1 mm/min F1. = 1 mm/min
Command range (mm/min)		0.001~1000000.000	0.0001 ~100000.0000

Inch input (inch)

Least input increment		(B) 0.0001 inch	(C) 0.00001 inch
F command increment (inch/min)	**without decimal point** **with decimal point**	F1 = 1 inch/min F1. = 1 inch/min	F1 = 1 inch/min F1. = 1 inch/min
Command range (inch/min)		0.0001~39370.0787	0.00001~3937.00787

- When commands without a decimal point have been assigned, it is not possible to assign commands under 1 mm/min (or 1 inch/min). To assign commands under 1 mm/min (or 1 inch/min), ensure that commands are assigned with a decimal point.
- The initial status after power-ON can be set to asynchronous feed (per-minute-feed) by setting the "Initial synchronous feed" parameter to OFF.

4.2.2 Feed per Revolution

M system : Δ **L system : O**

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 F command increment and command range are as follows.

[M system]

Metric input (mm)

Least input increment		(B) 0.001 mm	(C) 0.0001 mm
F command increment (mm/rev)	**without decimal point**	F1 = 0.01	F1 = 0.01
	with decimal point	F1. = 1	F1. = 1
Command range (mm/rev)		0.001~999.999	0.0001~99.9999

Inch input (inch)

Least input increment		(B) 0.0001 inch	(C) 0.00001 inch
F command increment (inch/rev)	**without decimal point**	F1 = 0.001	F1 = 0.001
	with decimal point	F1. = 1	F1. = 1
Command range (inch/rev)		0.0001~999.9999	0.00001~99.99999

- When commands without a decimal point have been assigned, it is not possible to assign commands under 1 mm/min (or 1 inch/min).
- The initial status after power-ON can be set to asynchronous feed (per-minute-feed) by setting the "Initial synchronous feed" parameter to OFF.
- The F command increments are common to all part systems.

[L system]

Metric input (mm)

Least input increment		(B) 0.001 mm	(C) 0.0001 mm
F command increment (mm/rev)	**without decimal point**	F1 = 0.0001	F1 = 0.0001
	with decimal point	F1. = 1	F1. = 1
Command range (mm/rev)		0.0001~999.999	0.00001~99.99999

Inch input (inch)

Least input increment		(B) 0.0001 inch	(C) 0.00001 inch
F command increment (inch/rev)	**without decimal point**	F1 = 0.000001	F1 = 0.000001
	with decimal point	F1. = 1	F1. = 1
Command range (inch/rev)		0.000001~99.999999	0.0000001~9.9999999

- When commands without a decimal point have been assigned, it is not possible to assign commands under 1 mm/min (or 1 inch/min).
- The initial status after power-ON can be set to asynchronous feed (per-minute-feed) by setting the "Initial synchronous feed" parameter to OFF.

4.2.4 F1-digit Feed

M system : O **L system : O**

When the "F1digt" parameter is ON, the feed rate registered by parameter in advance 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 feed rate set to support the code serves as the valid rate command. When a command higher than F5 is issued, it serves as a regular direct command with feed rate value of 5 digits following address F.
When an F1-digit command has been issued, the "In F1-digit" external output signal is output.

4.3 Override

4.3.1 Rapid Traverse Override

M system : O **L system : O**

(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.
Code method commands are assigned as combinations of bit signals from the PLC.

(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) Code method and value setting method can be selected by PLC processing.

4.3.2 Cutting Feed Override

M system : O **L system : O**

(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.
Code method commands are assigned as combinations of bit signals from the PLC.

(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.

4.3.3 2nd Cutting Feed Override

M system : O **L system : O**

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

4.3.4 Override Cancel

M system : O **L system : O**

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

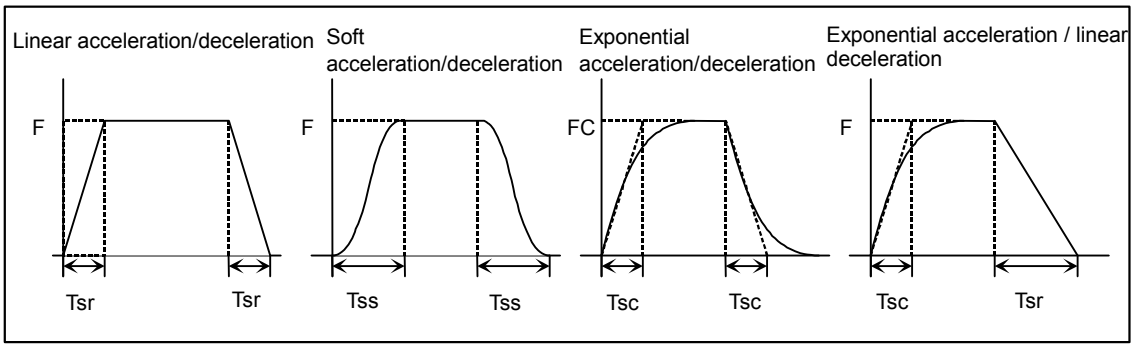
(Note 1) The override cancel signal is not valid for manual operation.
(Note 2) When the cutting feed override or second cutting feed override is 0%, the 0% override takes precedence and the override is not canceled.
(Note 3) The override cancel signal is not valid for rapid traverse.

4.4 Acceleration/Deceleration

4.4.1 Automatic Acceleration/Deceleration after Interpolation

M system : O **L system : O**

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 other that can be selected 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.

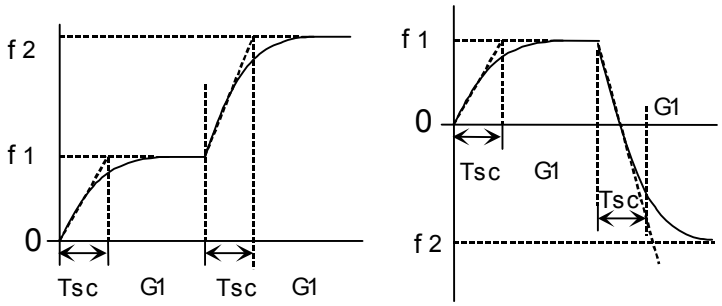


(Note 1) The rapid traverse feed acceleration/deceleration patterns are effective for the following: G00, G27, G28, G29, G30, rapid traverse feed in manual run, JOG, incremental feed, return to reference position.

(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).

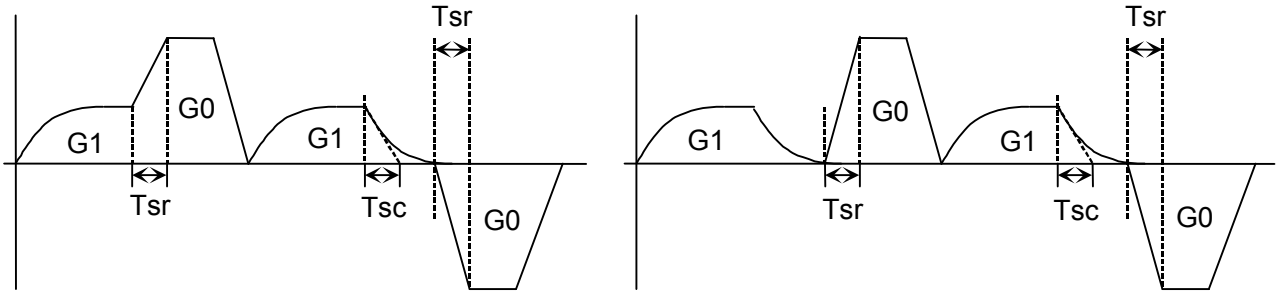
Acceleration/Deceleration during Continuing Blocks

(1) Continuous G1 blocks



The tool does not decelerate between blocks.

(2) Continuous G1-G0 blocks



If the G0 command direction is the same as that for G1, whether G1 is to be decelerated is selected using a parameter.
If no deceleration is set, superposition is performed even when G0 is in the constant inclination acceleration/deceleration state.
If the G0 command direction is the opposite of that for G1, G0 will be executed after G1 has decelerated.
(In the case of two or more simultaneous axes, G0 will also be executed after G1 has decelerated when the G0 command direction is the opposite of that for G1 for even one axis.)

4.4.2 Rapid Traverse Constant Inclination Acceleration/Deceleration

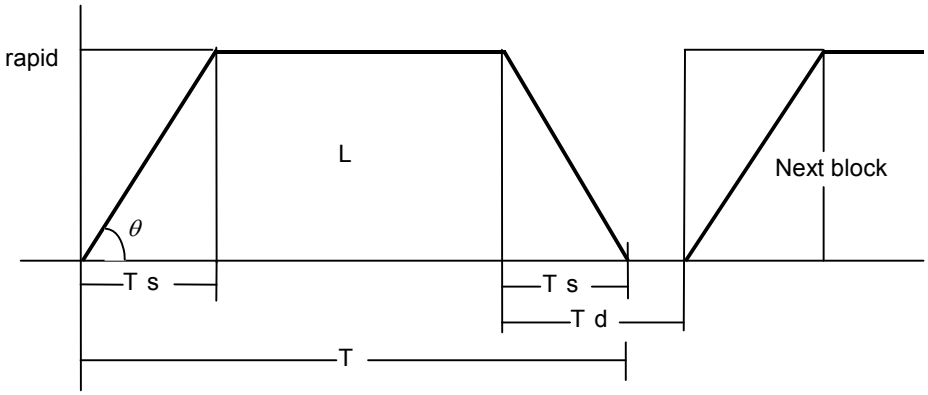
M system : O **L system : O**

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 longer than the acceleration and deceleration distance



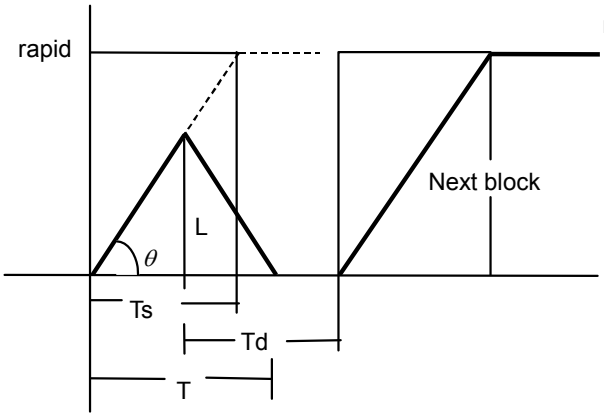
rapid : Rapid traverse rate
Ts : Acceleration/deceleration time constant
Td : Command deceleration check time
θ : Acceleration/deceleration inclination
T : Interpolation time
L : Interpolation distance

$$T = \frac{L}{rapid} + Ts$$

$$Td = Ts + (0 \sim 1.7\ ms)$$

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

(2) When the interpolation distance is shorter than the acceleration and deceleration distance



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

$$T = 2 \times \sqrt{Ts \times L / rapid}$$

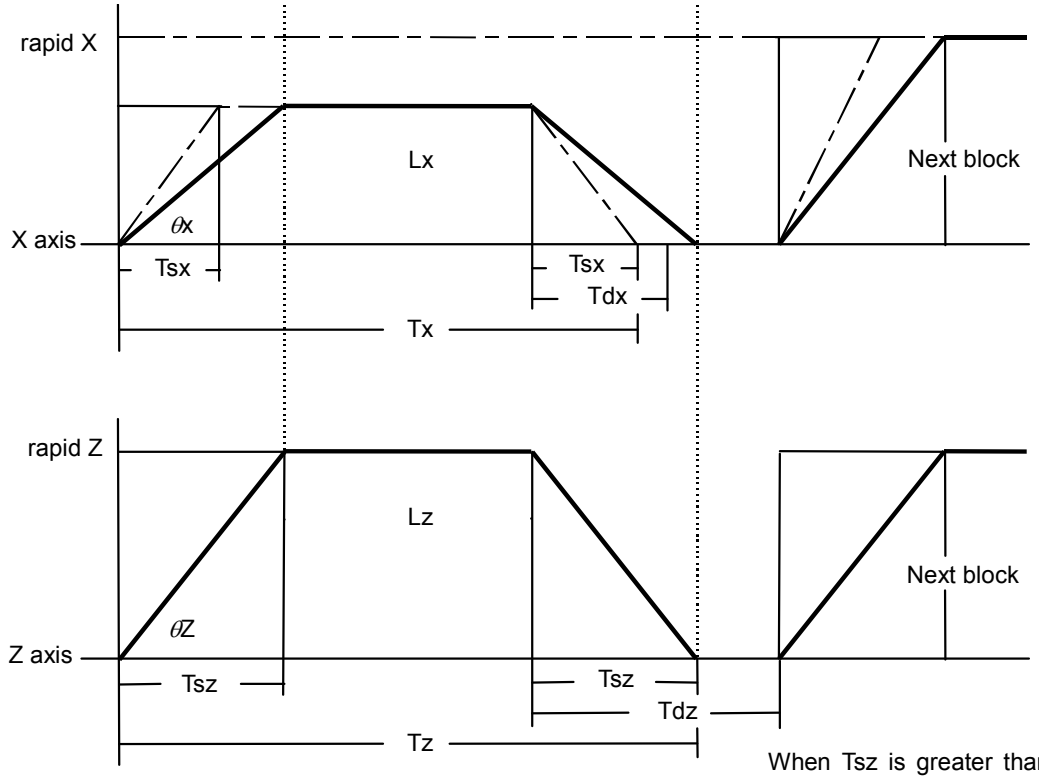
$$Td = \frac{T}{2} + (0 \sim 1.7\ ms)$$

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

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, Tsx < Tsz, and Lx ≠ Lz)

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 Tsz is greater than Tsx, Tdz is also greater than Tdx, and Td = Tdz in this block.

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).

4.5 Thread Cutting

4.5.1 Thread Cutting (Lead/Thread Number Designation)

M system : Δ **L system : ○**

The thread cutting with a 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 are performed based on the synchronization signals from the spindle encoder.

G33 Zz1/Ww1 Xx1/Uu1 Qq1 Ff1/Ee1 ;	
G33	: Thread 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)

The tables below indicate the thread lead ranges.

[M system]

Metric command		
Least input increment (mm)	**F (mm/rev)**	**E (mm/rev)**
0.001	0.001 to 999.999	0.00001 to 999.99999
0.0001	0.0001 to 99.9999	0.000001 to 99.999999

Inch command		
Least input increment (inch)	**F (inch/rev)**	**E (inch/rev)**
0.0001	0.0001 to 39.3700	0.000001 to 39.370078
0.00001	0.00001 to 3.93700	0.000001 to 3.937007

[L system]

Metric command		
Least input increment (mm)	**F (mm/rev)**	**E (mm/rev)**
0.001	0.0001 to 999.9999	0.00001 to 999.99999
0.0001	0.00001 to 99.99999	0.000001 to 99.999999

Inch command		
Least input increment (inch)	**F (inch/rev)**	**E (inch/rev)**
0.0001	0.000001 to 99.999999	0.000010 to 9.9999999
0.00001	0.0000001 to 9.9999999	0.00000001 to 0.99999999

The direction of the axis with a large movement serves as the reference for the lead.

(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

The tables below indicate the thread number.

[M system]

Metric command	
Least input increment (mm)	**Thread number command range (thread/inch)**
0.001	0.03 to 999.99
0.0001	0.255 to 9999.999

Inch command	
Least input increment (inch)	**Thread number command range (thread/inch)**
0.0001	0.0255 to 9999.9999
0.00001	0.25401 to 999.9999

[L system]

Metric command	
Least input increment (mm)	**Thread number command range (thread/inch)**
0.001	0.03 to 999.99
0.0001	0.255 to 9999.999

Inch command	
Least input increment (inch)	**Thread number command range (thread/inch)**
0.0001	0.0101 to 9999.9999
0.00001	0.10001 to 999.99999

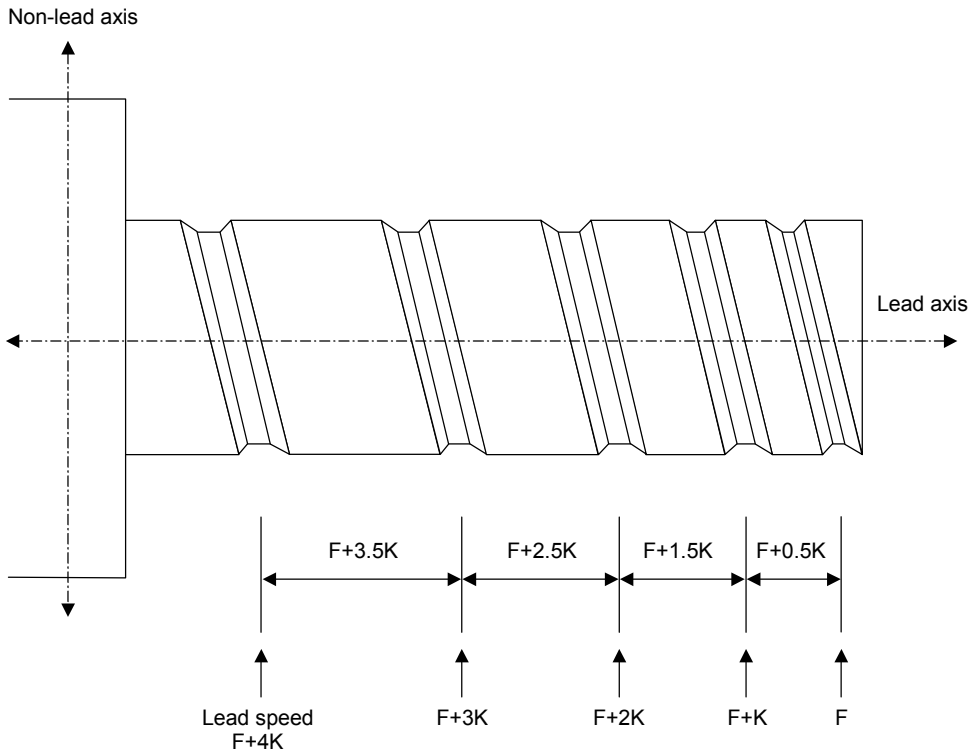
The number of thread per inch is commanded for both metric and inch systems, and the direction of the axis with a large movement serves as the reference.

4.5.2 Variable Lead Thread Cutting

M system : - **L system : O**

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

G34	**X/U__Z/W__F/E__K__;**
G34	: Variable lead thread cutting command
X/U	: Thread end point X coordinate
Z/W	: Thread end point Z coordinate
F/E	: Thread's basic lead
K	: Lead increment/decrement amount per thread rotation



4.5.3 Synchronous Tapping (with digital I/F spindle)

4.5.3.1 Synchronous Tapping Cycle

M system : Δ **L system : Δ**

This function performs tapping through synchronized control of the 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 Pp1 Ff1 Ss1 , R1 ;	
G84	: Synchronous tapping mode ON, forward tapping
G74	: Synchronous tapping mode ON, reverse tapping
Xx1, Yy1	: Hole position data, hole drilling coordinate position
Zz1	: Hole machining data, hole bottom position
Rr1	: Hole machining data, hole R position
Pp1	: Hole machining data, dwell time at hole bottom
Ff1	: Z-axis feed amount (tapping pitch) per spindle rotation
Ss1	: Spindle speed
,R1	: Synchronous system selection

(2) Tapping thread number assignment

G84(G74) Xx1 Yy1 Zz1 Rr1 Pp1 Ee1 Ss1 , R1 ;	
G84	: Synchronous tapping mode ON, forward tapping
G74	: Synchronous tapping mode ON, reverse tapping
Xx1, Yy1	: Hole position data, hole drilling coordinate position
Zz1	: Hole machining data, hole bottom position
Rr1	: Hole machining data, hole R position
Pp1	: Hole machining data, dwell time at hole bottom
Ee1	: Tap thread number per 1-inch feed of Z axis
Ss1	: Spindle speed
,R1	: Synchronous system selection

The control state will be as described below when a tapping mode command (G74, G84) is commanded.

1. Cutting override Fixed to 100%
2. Feed hold invalid
3. "In tapping mode" signal is output
4. Deceleration command between blocks invalid
5. Single block invalid

The tapping mode will be canceled with the following G commands.

G61..... Exact stop check mode
G61.1..... High-accuracy control mode
G62..... Automatic corner override
G64..... Cutting mode

4.5.3.2 Pecking Tapping Cycle

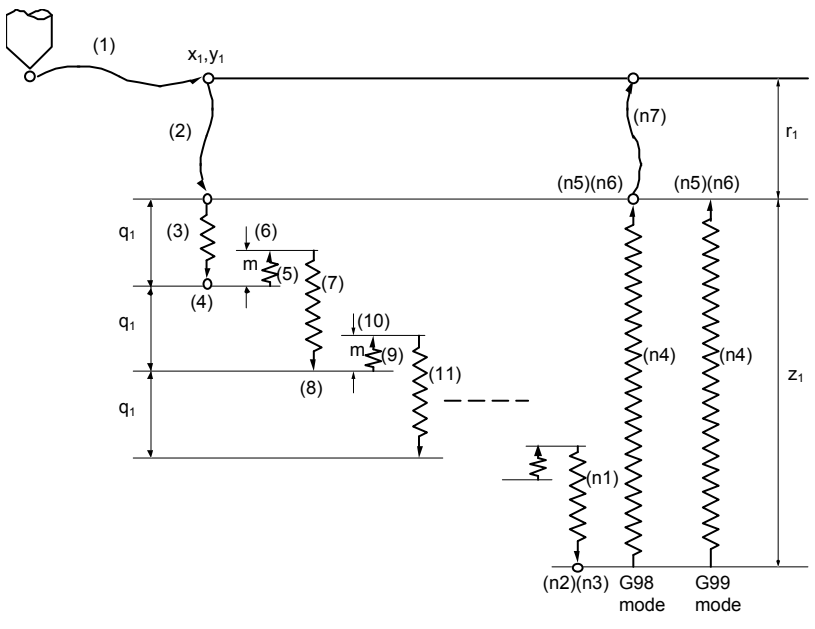
M system : Δ **L system : -**

This function performs cutting the workpiece to the hole bottom for a multiple number of passes by designating the depth of cut per pass. The load applied to the tool can be reduced.
The amount retracted from the hole bottom is set to the parameters.
When the pecking tapping cycle is executed in the synchronous tapping mode, the synchronous tapping cycle option and pecking tapping cycle option are required.
When "depth of cut per pass Q" is designated in the block containing the G84 or G74 command in the state where the pecking tapping cycle is selected by parameter, the pecking tapping cycle is executed.
In the following cases, the normal tapping cycle is established.

- When Q is not designated
- When the command value of Q is zero

G84(G74) Xx1 Yy1 Zz1 Rr1 Qq1 Ff1 Ee1 Pp1 Ss1 ,Ss2 ,Ii1 ,Jj1 ,Rr2 ;	
G84	: G84 forward tapping cycle
G74	: G74 reverse tapping cycle
Xx1, Yy1	: Hole drilling position
Zz1	: Hole bottom position
Rr1	: Point R position
Qq1	: Depth of cut per pass (designated as an incremental position)
Ff1	: Z-axis feed amount (tapping pitch) per spindle rotation
Ee1	: Tap thread number per 1-inch feed of Z axis
Pp1	: Dwell time at hole bottom position
Ss1	: Rotation speed of spindle
, Ss2	: Rotation speed of spindle during retract
, Ii1	: In-position width of positioning axis
, Jj1	: In-position width of hole drilling axis
, Rr2	: Synchronization method selection (r2=1 synchronous, r2=0 asynchronous)

(Note 1) When ",R0" is commanded, F address is regarded as cutting feedrate.



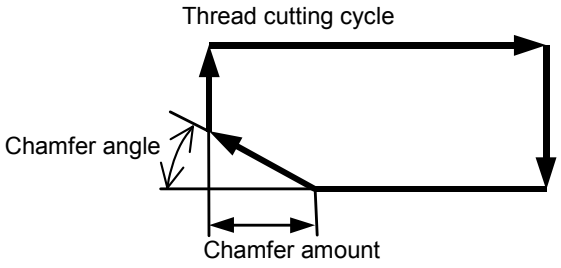
(1)	G0 Xx1 Yy1 ,Ii1
(2)	G0 Zr1
(3)	G1 Zq1 Ff1
(4)	M4 (Spindle reverse rotation)
(5)	G1 Z-m Ff1
(6)	M3 (Spindle forward rotation)
(7)	G1 Z(q1+m) Ff1
(8)	M4 (Spindle reverse rotation)
(9)	G1 Z-m Ff1
(10)	M3 (Spindle forward rotation)
(11)	G1 Z(q1+m) Ff1
:	:
(n1)	G1 Z(z1-q1*n) Ff1
(n2)	G4 Pp1
(n3)	M4 (Spindle reverse rotation)
(n4)	G1 Z-z1 Ff1 Ss2
(n5)	G4 Pp1
(n6)	M3 (Spindle forward rotation)
(n7)	G98 mode G0 Z-r1 ,Ij1 G99 mode No movement

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

4.5.4 Chamfering

M system : - **L system : O**

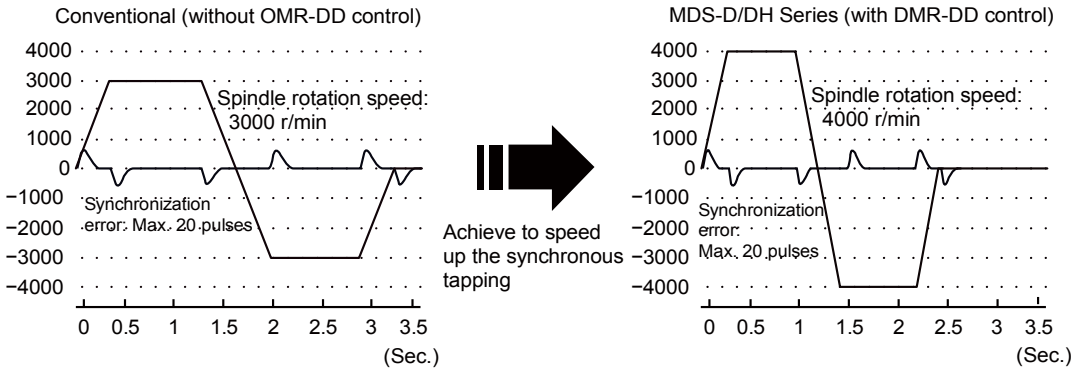
Chamfering can be validated during the thread cutting cycle by using external signals.
The chamfer amount and angle are designated with parameters.



4.5.8 High-speed Synchronous Tapping(OMR-DD)

M system : Δ **L system : Δ**

The servo axis directly detects and compensates the spindle's delay in tracking by using drive unit communication over the high-speed optical servo network. By minimizing the synchronization error, the accuracy of the synchronous tapping is increased. If the degree of accuracy is same as conventional one, the spindle rotation speed is increased and the cycle time of the synchronous tapping is shortened.

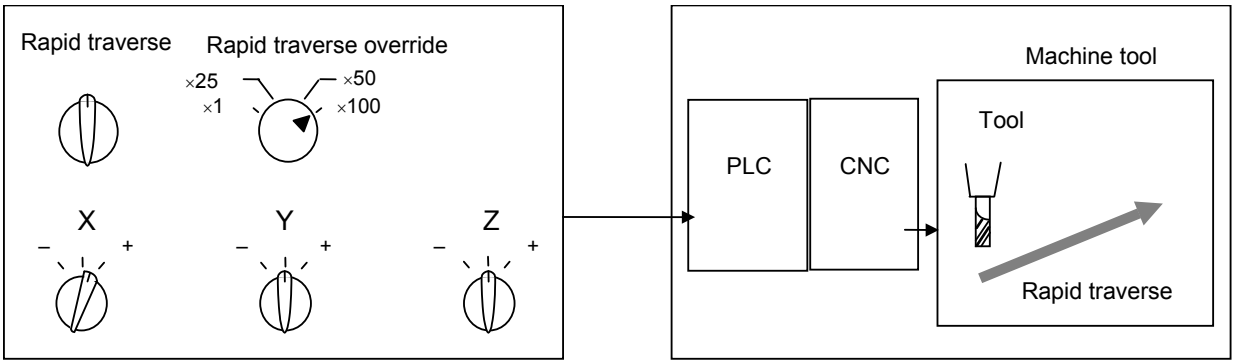


4.6 Manual Feed

4.6.1 Manual Rapid Traverse

M system : O **L system : O**

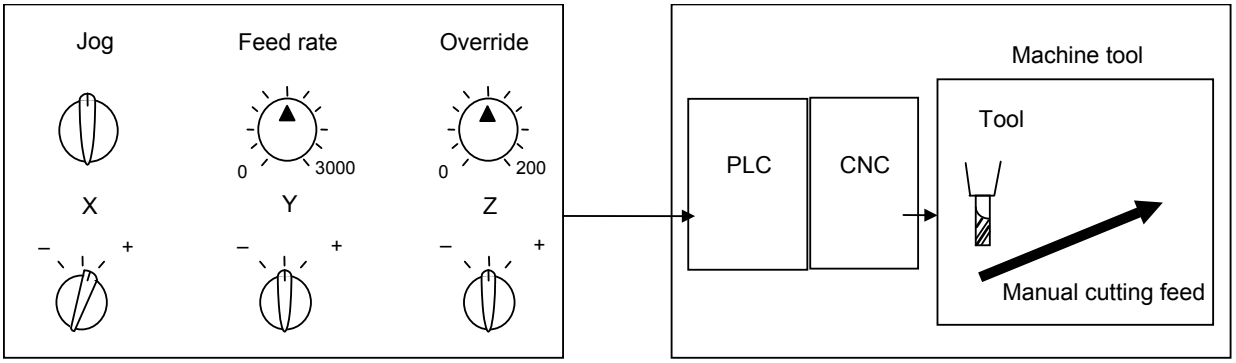
When the manual rapid traverse mode is selected, the tool can be moved at the rapid traverse rate for each axis separately. Override can also be applied to the rapid traverse rate by means of the rapid traverse override function.
Rapid traverse override is common to all part systems.



4.6.2 Jog Feed

M system : O **L system : O**

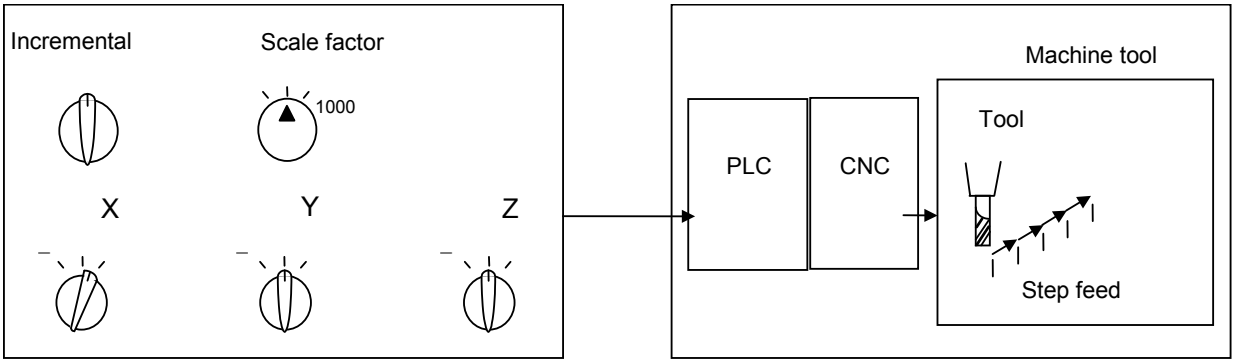
When the jog feed mode is selected, the tool can be moved in the axis direction (+ or –) in which the machine is to be moved at the per-minute feedrate. The jog feed rate is common to all part systems.



4.6.3 Incremental Feed

M system : O **L system : O**

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 input increment that was set with the parameter by the incremental feed magnification rate.
The incremental feed amount parameter and its magnification rate are common to all part systems.



4.6.4 Handle Feed

M system : Δ **L system : Δ**

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

(3-axes)
In the handle feed mode, 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.

4.7 Dwell

4.7.1 Dwell (Time-based Designation)

M system : O **L system : O**

The G04 command temporarily stops machine movement and sets the machine stand-by 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 G04 Pp1 ;		
G94	:	Asynchronous
G04	:	Dwell
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.)

5. Program Memory/Editing

5.1 Memory Capacity

Machining programs are stored in the CNC memory.

5.1.1 Memory Capacity (Number of Programs Stored)

(Note 1) The tape length will be the total of two part systems when using the 2-part system specifications.

5.1.1.1 15KB [40m] (64 programs)

M system : ○ **L system : ○**

5.1.1.2 30KB [80m] (128 programs)

M system : Δ **L system : Δ**

5.1.1.3 60KB [160m] (200 programs)

M system : Δ **L system : Δ**

5.1.1.4 125KB [320m] (200 programs)

M system : Δ **L system : Δ**

5.1.1.5 230KB [600m] (400 programs)

M system : Δ **L system : Δ**

5.1.1.6 500KB [1280m] (1000 programs)

M system : Δ **L system : Δ**

5.1.1.7 1000kB[2560m] (1000 programs)

M system : Δ **L system : Δ**

5.1.1.8 2000kB[5120m] (1000 programs)

M system : Δ **L system : Δ**

5.2 Editing

5.2.1 Program Editing

M system : O **L system : O**

The following editing functions are possible.

(1) Program erasing
  (a) Machining programs can be erased individually or totally.
  (b) When all machining programs are to be erased, the programs are classified with their No. into B: 8000 to 8999, C: 9000 to 9999, and A: all others.

(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.

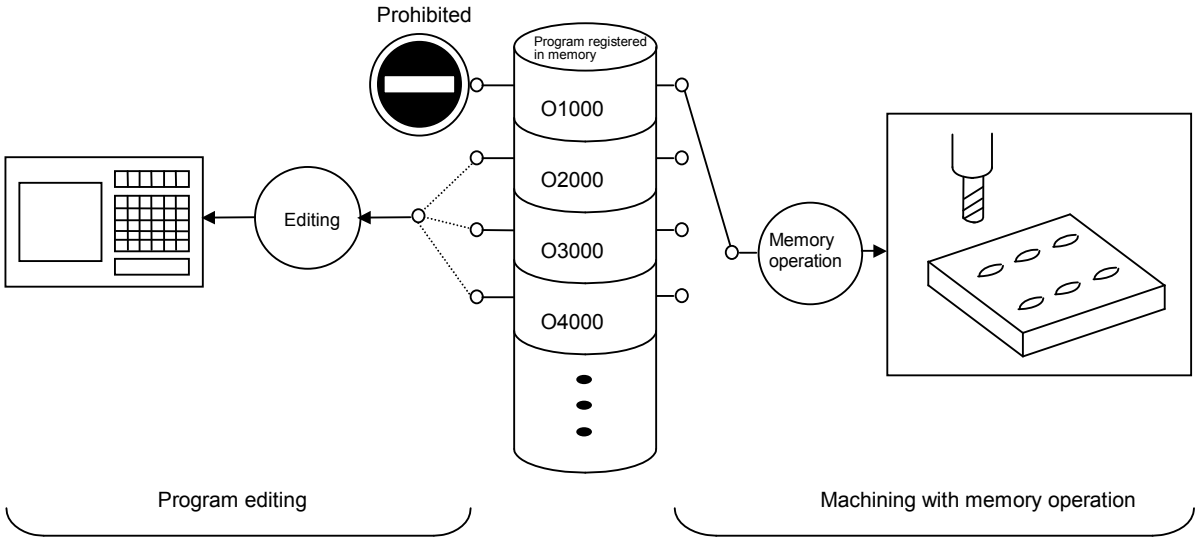
(3) Program copying
  (a) Machining programs stored in the controller memory can be copied, condensed or merged.
  (b) The program No. of the machining programs in the memory can be changed.

(4) Program editing
  (a) Overwriting, inserting and erasing can be done per character.

5.2.2 Background Editing

M system : O **L system : O**

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



(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.

5.2.4 Word Editing

M system : O **L system : O**

This function enables to edit programs in word unit by insertion, deletion, and replacement.

6. Operation and Display

6.1 Structure of Operation/Display Panel

6.1.2 Color Display (GOT)

M system : ☐ **L system : ☐**

Choose GOT (Mitsubishi Graphic Operation Terminal) from listed below.

Model	Type of power supply	Model name	Screen size (resolution)			
			15-type XGA [1024x768]	12.1-type SVGA [800x600]	10.4-type SVGA [800x600]	8.4-type SVGA [800x600]
GT16	100-240VAC	GT1695M-XTBA	☐	-	-	-
		GT1685M-STBA	-	☐	-	-
		GT1675M-STBA	-	-	☐	-
		GT1665M-STBA	-	-	-	☐
	24VDC	GT1695M-XTBD	☐	-	-	-
		GT1685M-STBD	-	☐	-	-
		GT1675M-STBD	-	-	☐	-
		GT1665M-STBD	-	-	-	☐
GT15	100-240VAC	GT1595-XTBA	☐	-	-	-
		GT1585V-STBA	-	☐	-	-
		GT1585-STBA	-	☐	-	-
		GT1575V-STBA	-	-	☐	-
		GT1575-STBA			☐	
	24VDC	GT1595-XTBD	☐	-	-	-
		GT1585V-STBD	-	☐	-	-
		GT1585-STBD	-	☐	-	-
		GT1575V-STBD	-	-	☐	-
		GT1575-STBD	-	-	☐	-

TFT color liquid crystal display (High brightness, wide angle view), the number of displayed colors: 65535 colors.
Standard memory: GT16 15MB, GT15 9MB
All types of GT16 support multi-media and video/RGB, and video/RGB is supported for GT15 if the model name is GT15□5V-STB□.

6.2 Operation Methods and Functions

6.2.2 Absolute Value/Incremental Value Setting

M system : ○ **L system : ○**

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.

- Tool compensation amount screen
- Coordinate system offset screen

6.2.3 Single-NC and Multi-display Unit Switch

M system : ○ **L system : ○**

By adding an Ethernet hub, up to eight displays can be changed over for one CNC. (Note that the max. number of displays that can be connected is limited by the machine operation panel specifications.)

6.2.4 Multi-NC and Common-display Unit

M system : ○ **L system : ○**

By adding an Ethernet hub, up to 64 CNC modules can be changed over and displayed on one display. (Note that the max. number of modules that can be connected is limited by the machine operation panel specifications.)

6.2.5 Displayed Part System Switch

M system : ○ **L system : ○**

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

6.2.10 Screen Saver, Backlight OFF

M system : ○ **L system : ○**

The GOT's screensaver function protects the display by turning the backlight OFF after the preset time has elapsed.

6.2.15 Screen Capture

M system : ○ **L system : ○**

The GOT's hard copy function captures the screen image in JPEG or bitmap format.

6.3 Display Methods and Contents (CNC Monitor Function)

6.3.1 Status Display

M system : ○ **L system : ○**

The status of the program currently being executed is indicated.
(1) Display of G, S, T, M commands and 2nd miscellaneous command modal values
(2) Feed rate display
(3) Tool offset number and offset amount display
(4) Real speed display (Note 1)

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

6.3.2 Clock Display

M system : ○ **L system : ○**

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

6.3.3 Position Display

M system : ○ **L system : ○**

Various information related to operation, such as the axis counter, speed display and MSTB command are displayed on the Position Display screen. The following operations regarding operation can be executed.
(1) Operation search
(2) Setting of common variables
(3) Setting of local variables
(4) Counter zero
(5) Origin zero
(6) Manual numeric command, etc.

6.3.4 Tool Compensation/Parameter

M system : ○ **L system : ○**

Tool/workpiece related settings, user parameter settings, manual numeric command issuing and tool length measurements can be carried out on the Tool Compensation/Parameter screen.

6.3.5 Program

M system : ○ **L system : ○**

Machining program searching, creating and editing (addition, deletion, change), program list display and MDI editing can be carried out on the Program screen.

6.3.6 Alarm Diagnosis

M system : ○ **L system : ○**

The following operations related to CNC diagnosis can be carried out on the Diagnosis screen.
(1) Display of hardware, software and drive unit configuration
(2) Operation monitor of servo and spindle drive unit
(3) Diagnosis of NC input/output signal (interface diagnosis)
(4) Display of operation history
(5) Display of alarm / stop code history list
(6) Data sampling for maintenance
(7) Deleting, copying and list displaying of machining program

6.3.8 Additional Languages

6.3.8.1 Japanese

M system : ○ **L system : ○**

6.3.8.2 English

M system : ○ **L system : ○**

6.3.8.6 Spanish

M system : Δ **L system : Δ**

6.3.8.7 Chinese

6.3.8.7.2 Simplified Chinese Characters

M system : Δ **L system : Δ**

6.3.8.14 Polish

M system : Δ **L system : Δ**

7. Input/Output Functions and Devices

7.1 Input/Output Data

CNC data input/output function of GOT (Mitsubishi Graphic Operation Terminal) is used.

- GT15: Compact flash
- GT16: Compact flash for USB memory

Various data of CNC can be input/output for the following memory card which is attached to GOT.
- GT15: Compact flash
- GT16: Compact flash or USB memory

7.1.1 Machining program input / output

M system : O **L system : O**

7.1.2 Tool offset data input / output

M system : O **L system : O**

7.1.3 Common variable input / output

M system : O **L system : O**

7.1.4 Parameter input / output

M system : O **L system : O**

7.1.5 History data output

M system : O **L system : O**

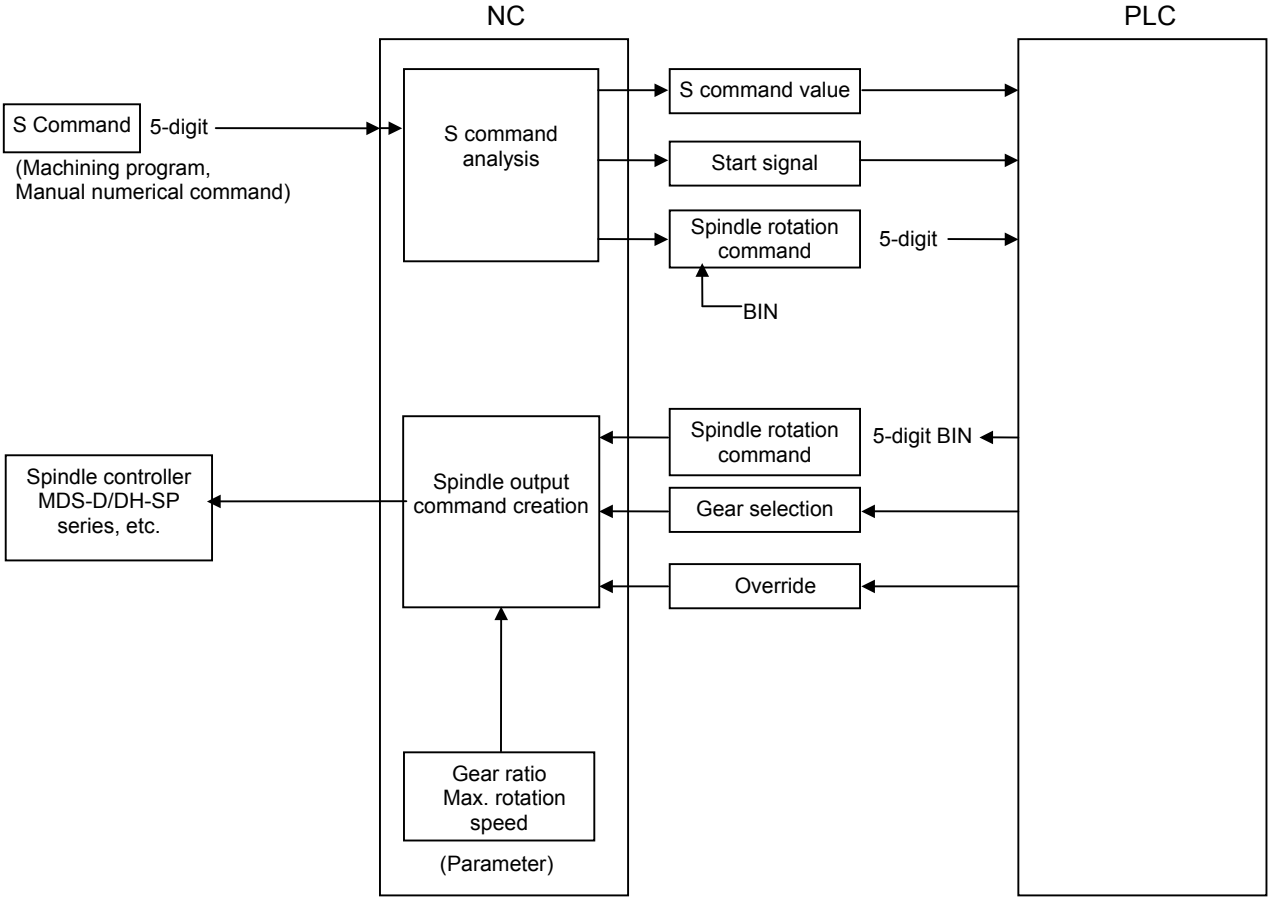
8. Spindle, Tool and Miscellaneous Functions

8.1 Spindle Functions (S)

8.1.1 Spindle Control Functions

The spindle rotation speed is determined in consideration of the override and gear ratio for the S command given 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 (S-99999999 to S99999999) is commanded, a signed 32-bit binary data and start signal will be output to the PLC.
When multiple spindle control "Sn = ****" method, up to seven sets 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. 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.

(Note 1) S command can be commanded by eight digits. However, setting range of the parameter highest rotation speed and rotation speed limit, etc. are five digits or less. So, S command which can be substantially controlled are five digits or less.
(Note 2) The display of S command is five digits or less display on some screens.

8.1.1.1 Spindle Digital I/F

M system : ○ **L system : ○**

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

8.1.1.2 Spindle analog I/F

M system : Δ (using MELSEC I/O) **L system : Δ (using MELSEC I/O)**

Spindle control can be performed with analog voltage input type spindle instead of digital spindle.

8.1.1.3 Coil Switch

M system : ○ **L system : ○**

Constant output characteristics can be achieved across a broad spectrums down to the low-speed ranges 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 : ○ **L system : ○**

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

8.1.2 S Code Output

M system : ○ **L system : ○**

When an 8-digit number following address S (S-99999999 to S99999999) is commanded, a 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 process (presence of DEN signal process), 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.

(Note) The display of S command is five digits or less display on some screens.

8.1.3 Constant Surface Speed Control

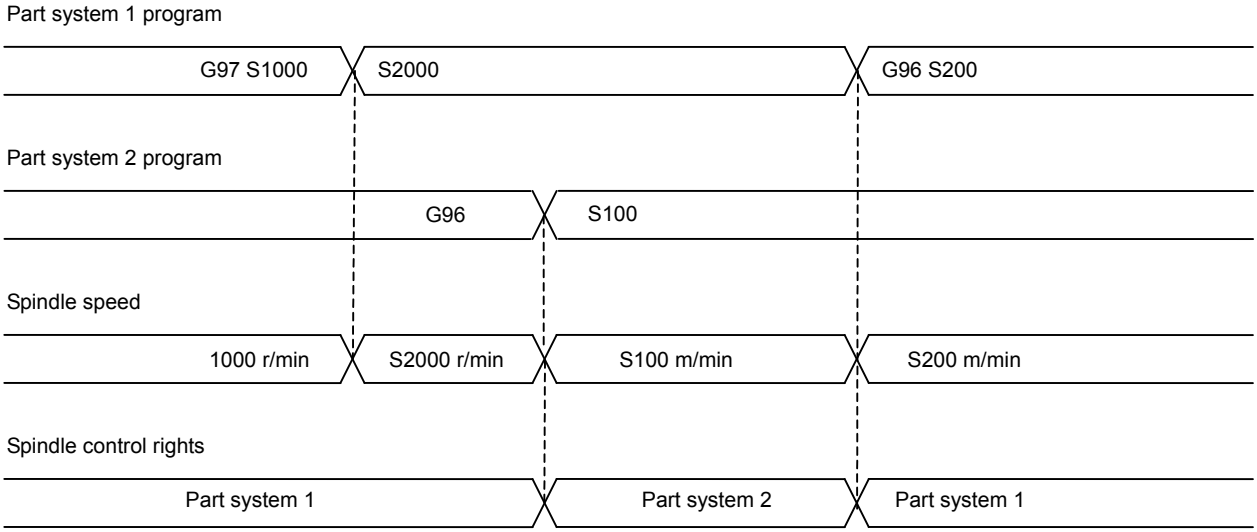
M system : Δ **L system : Δ**

With radial direction cutting, this function enables the spindle speed to be changed in accordance with changes in the radial direction coordinate values 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) If there is only one spindle, the spindle will not operate normally if the constant surface speed control command, S command or spindle related M command is commanded randomly from each part system. These commands must be commanded from only one certain part system, or commanded simultaneously with standby.
The controller will execute the following control for the constant surface speed control and S commands. The part system from which an S command was issued last will have the spindle control rights. That part system will judge whether the constant surface speed command mode is valid or canceled, and will execute spindle control.



8.1.4 Spindle Override

M system : O **L system : O**

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

(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) 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%.

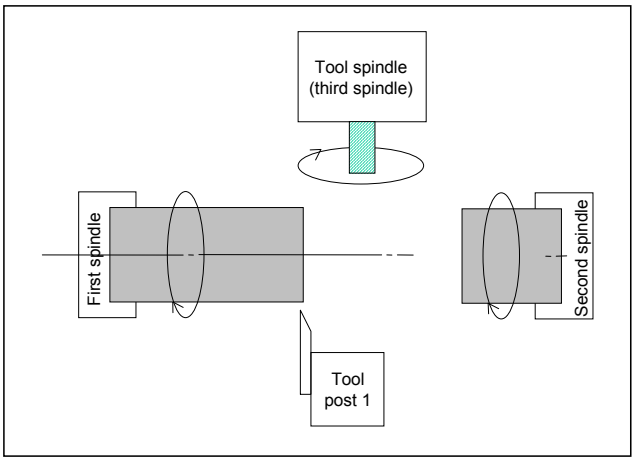
(Note 1) Selection between code method and value setting method can be designated by user PLC processing.

8.1.5 Multiple-spindle Control

When using a machine tool equipped with several spindles (up to seven spindles), this function controls those spindles.

Multiple-spindle control I: Control based on a spindle selection command, such as G43.1, and spindle control command [S******;] or [SO=******;], etc.

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



8.1.5.1 Multiple-spindle Control I

M system : O **L system : O**

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

Command format

G43.1;	Selected spindle control mode ON; the selected spindle number is set using a parameter.
G44.1;	Second spindle control mode ON

(2) Spindle control commands (using an extended word address (SO=******))
In addition to using the "S******" S commands, it is also possible to assign commands which differentiate the applicable spindle among the first through seventh spindles by using the SO=******.
The S command can be issued from a machining program for any part system.
The number of spindle axes differs according to the model, so check the specifications.
The C6 T and L System and C64 T System cannot control multiple spindles in one part system.

Command format

SO=****;**

O : Number assigned as the spindle number (1: first spindle; 2: second spindle; ··· 7: seventh spindle); variables can be designated.

****** : Rotational speed or surface speed value assigned by 6-digit analog command; variables can be designated.

8.1.6 Spindle Orientation

M system : O **L system : O**

This function stops the spindle rotation at a certain position.

(1) Orientation
This function stops the spindle rotation at a certain position when using the digital spindle.
When the orientation command is used, the spindle will rotate several times and then stop at the orientation point. The orientation point is the Z-phase position when using encoder orientation (PLG and external encoder) or the proximity switch neighborhood when using the proximity switch method.

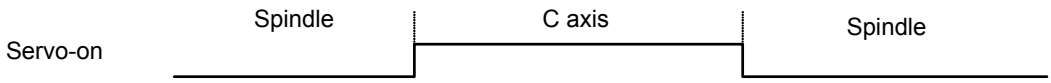
(2) Multi-point orientation
This function performs orientation to a position other than the Z-phase position by inputting a shift amount with the parameter or PLC. The shift amount is 0 to 35999. (Unit: 360°/36000 = 0.01°)

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

8.1.7 Spindle position control (Spindle/C axis control)

M system : Δ **L system : Δ**

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.



Servo-off ---------- Spindle (C axis cannot be controlled)
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.)

8.1.8 Spindle Synchronization

In a machine with two or more spindles, this function controls the rotation speed and phase of one selected spindle (synchronized spindle) in synchronization with the rotation of the other selected spindle (basic spindle). There are two methods for giving commands: G code and PLC.
It is used in cases where, for instance, workpiece clamped to the basic spindle is to be clamped to the synchronized spindle instead or where the spindle rotation speed is to be changed while one workpiece remains clamped to both spindles.

8.1.8.1 Spindle Synchronization I

M system : Δ **L system : Δ**

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

Command format

Spindle synchronization control cancel (G113)
This command releases the state of synchronization between two spindles whose rotation has been synchronized by the spindle synchronization command.

G113;

Spindle synchronization control ON (G114.1)
This command is used to designate the basic spindle and the spindle to be synchronized with the basic spindle, and it places the two designated spindles in the synchronized state.
By designating the synchronized spindle phase shift amount, the phases of the basic spindle and synchronized spindle can be aligned.

G114.1 H__ D__ R__ A__ ;
H__ : Selects the basic spindle.
D__ : Selects the spindle to be synchronized with the basic spindle.
E__ : Designates the synchronized spindle phase shift amount.
A__ : Designates the spindle synchronization acceleration/deceleration time constant.

8.1.8.2 Spindle Synchronization II

M system : Δ **L system : Δ**

Whereas the spindle synchronization I executes the selection of the spindles to be synchronized, the start of the synchronization and other settings with G code in the machining program, this function designates all these from the PLC.

The spindle synchronization control mode is established by inputting the spindle synchronization control signal. While this mode is established, the synchronized spindle is controlled in synchronization with the rotation speed assigned for the basic spindle.

8.1.11 Spindle Speed Clamp

M system : O **L system : O**

The spindle rotation speed is clamped between maximum rotation speed and minimum rotation speed.

8.2 Tool Functions (T)

8.2.1 Tool Functions (T Command)

M system : O **L system : O**

This function commands the tool No. by an 8-digit number that follows address T (T0 to T99999999). Tool compensation No.(tool length compensation and/or tool nose wear compensation) will be displayed for L system.

(1) M system

When an 8-digit number following address T (T00000000 - T99999999) is assigned, 8-digit code data and start signal will be output to PLC.

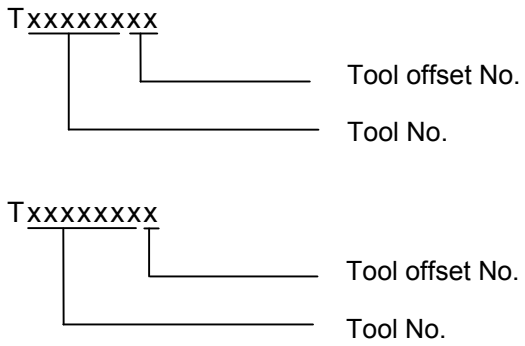
Only one set of T commands can be commanded in a block.

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

(Note 1) There are some screens in the setting and display unit that cannot display all eight digits.

(2) L system

The command is issued with an 8-digit number following address T (T0 - T99999999). The high-order 6 digits or 7 digits are designated as the tool No., and the low-order 2 digits or 1 digit are designated as the offset No. Which method is to be used is designated with parameters.



The 6-digit (or 7-digit) tool No. code data and start signal will be output to the PLC.
Processing and complete sequences must be incorporated on the PLC side for all T commands.

(Note 1) There are some screens in the setting and display unit that cannot display all eight digits.

8.3 Miscellaneous Functions (M)

8.3.1 Miscellaneous Functions

M system : O **L system : O**

When an 8-digit number (M00000000~M99999999) is assigned following address M, the 8-digit code data and start signal are output to the PLC.

Apart from the above signals, various special independent signals are also output for the following signals.

M00 : Program stop
M01 : Optional stop
M02 : Program end
M30 : Program end

Respective processing and complete sequences must be incorporated on the PLC side for all M commands from M00000000 to M99999999.
M98 and M99 have specific purposes and can not be used.

(Note 1) There are some screens in the setting and display unit that cannot display all eight digits.

8.3.2 Multiple M Codes in 1 Block

M system : O **L system : O**

Four sets of M commands can be issued simultaneously in a block.
Respective processing and completion sequences are required for all M commands included in a block (except M98 and M99).

(Note 1) 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 : O **L system : O**

When the M00, M01, M02 or M30 command is assigned during an automatic operation (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

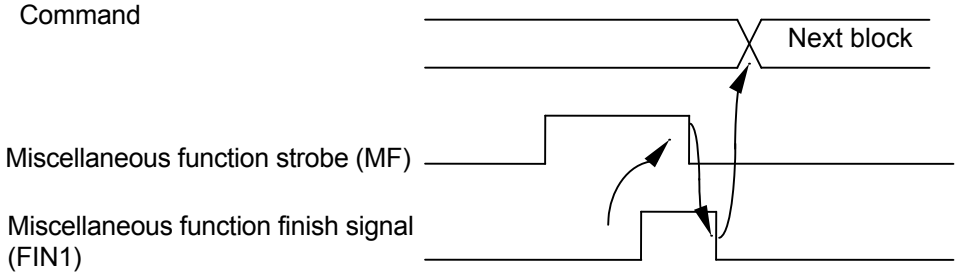
If movement or dwell command exists in the same block as these M commands, this signal is output upon completion of the movement or dwell command.

8.3.4 Miscellaneous Function Finish

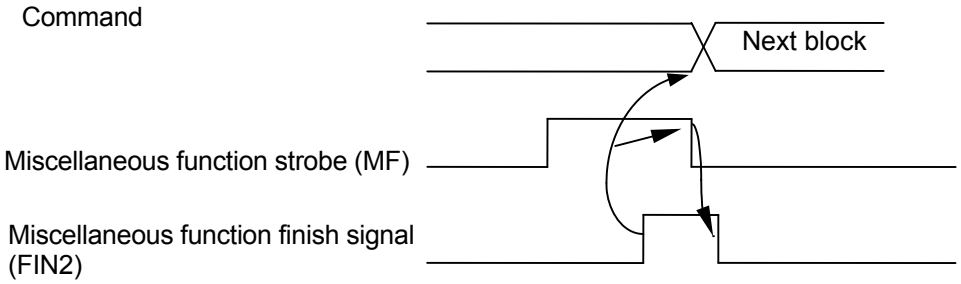
M system : O **L system : O**

These signals inform the CNC system that a miscellaneous function (M), spindle function (S), tool function (T) or 2nd miscellaneous function (A, B, C) has been assigned, and that the PLC which has received it has completed the required operation. They include miscellaneous function finish signal 1 (FIN1) and miscellaneous function finish signal 2 (FIN2).

Miscellaneous function finish signal 1 (FIN1)
When the controller checks that FIN1 is ON, it sets the function strobes OFF. Furthermore, when the PLC checks that the function strobes are OFF, it sets FIN1 OFF. The controller checks that FIN1 is OFF and advances to the next block.
Below is an example of a time chart applying when a miscellaneous function has been assigned.



Miscellaneous function finish signal 2 (FIN2)
When the controller checks that FIN2 is ON, it sets the function strobes OFF and simultaneously advances to the next block. The PLC checks that the strobe signals are OFF and sets FIN2 OFF.
Below is an example of a time chart applying when a miscellaneous function has been assigned.



8.4 2nd Miscellaneous Functions (B)

8.4.1 2nd Miscellaneous Functions

M system : ○ **L system : ○**

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

9. Tool Compensation

9.1 Tool Length/Tool Position

9.1.1 Tool Length Compensation

M system : O **L system : O**

These commands make it possible to control the axis movement by offsetting the position of the end point of the movement command by the amount set on the TOOL OFFSET screen.
Using this function, it is possible to offset 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	;	Tool length offset can be provided not only for the Z axis but for all other axes which can be controlled in the system (X, Y, etc.).
G44	Zz1	Hh1	;	
Offset direction	Offset axis	Offset No.		
G49 ;				Tool length offset cancel

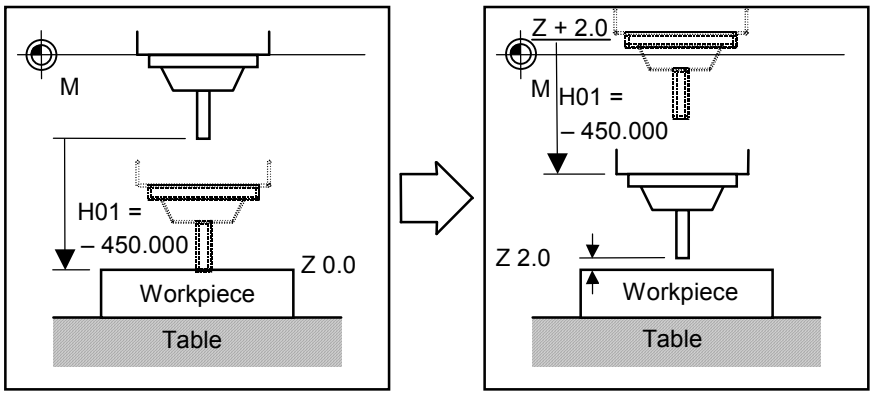
The offset direction is determined by the G command.
G43: Forward direction (z1 + h1)
G44: Reverse direction (z1 – h1)
Offset can be canceled by the following G commands.
G49;
G43 H0;
G44 H0;

(Note 1) When the tool length offset axis is returned to the reference point, the offset of that axis is canceled.

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

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

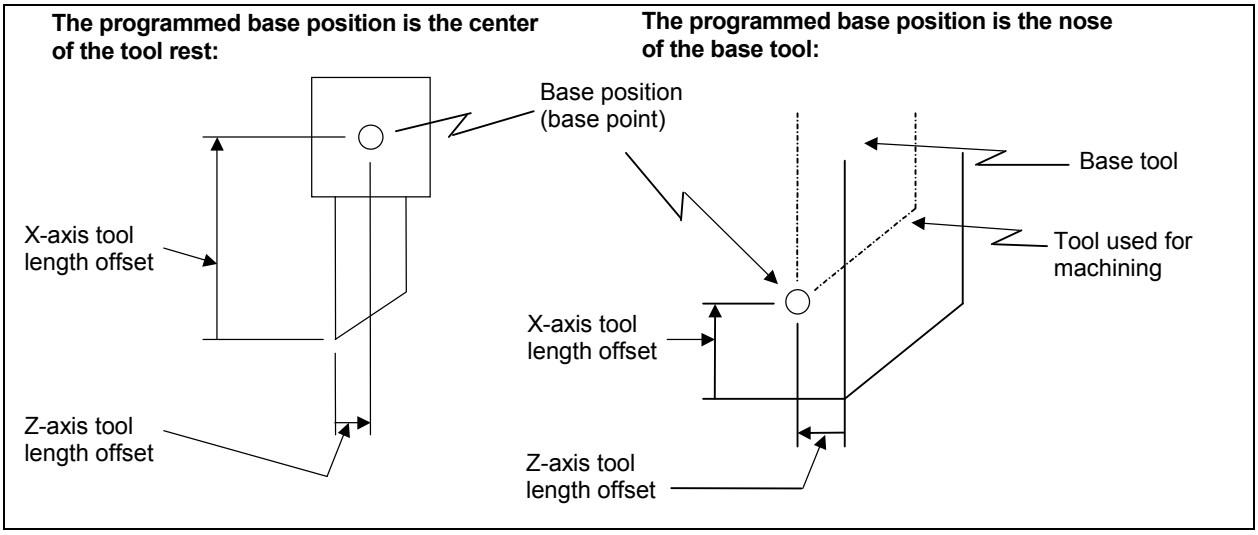
(Note) The tool length offset amount is set as a negative value such as H01 = –450.000.



(2) L system

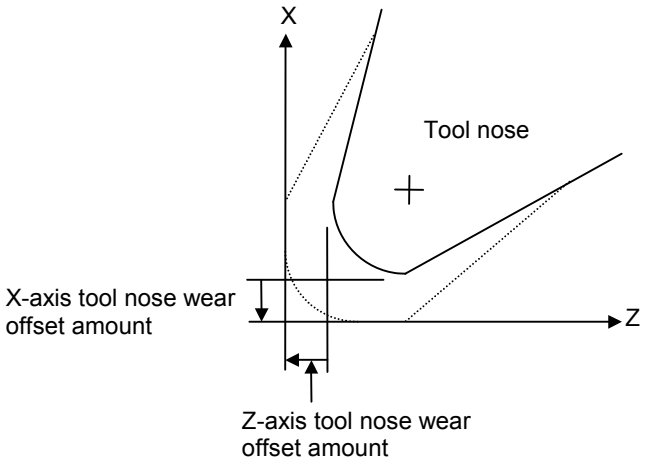
(a) Shape offset

Tool length is offset 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.



(b) Wear offset

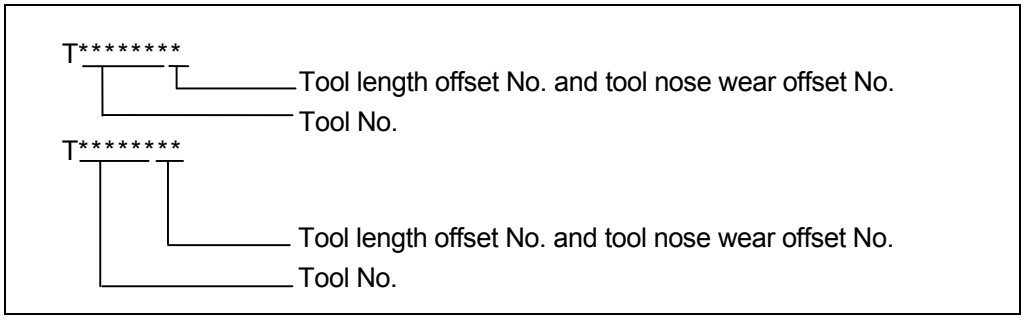
The wear of a tool nose can be offset.



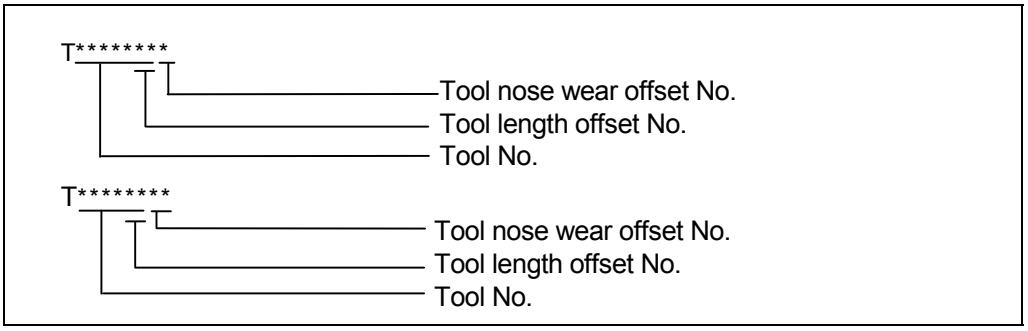
(c) Command format

Tool offset is performed by a T command. It is specified in eight digits following address T. Tool offset is divided into two types: tool length offset and tool nose wear offset. The Nos. of such two types of offsets are specified by a parameter. Also a parameter is used to specify whether the offset Nos. is specified by one or two low-order digits of a T command.

1. Specifying tool length and wear offset Nos. together using one or two low-order digits of the T command



2. Specifying tool length and wear offset Nos. separately



The tool offset for the L system is valid only for the X and Z axes.

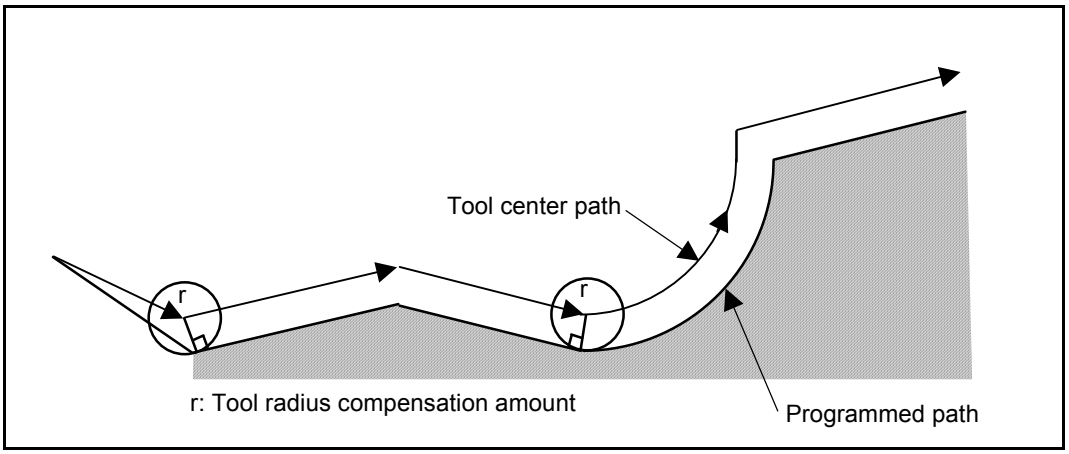
9.2 Tool Radius

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 change during tool radius compensation
G39	Corner arc during tool radius compensation
G40	Tool radius compensation cancel
G41	Tool radius compensation left command
G42	Tool radius compensation right command



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

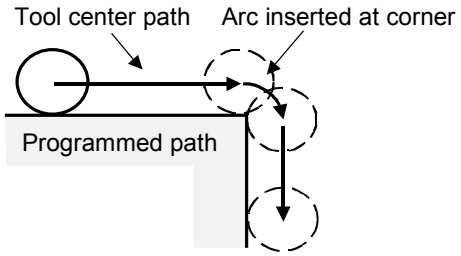
G17 : Compensation plane
G01 : Cutting command
G41 : Left compensation
Xx1,Yy1 : Movement axis
Dd1 : Compensation No.

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 ;	
Xx1, Yy1	: Movement amount



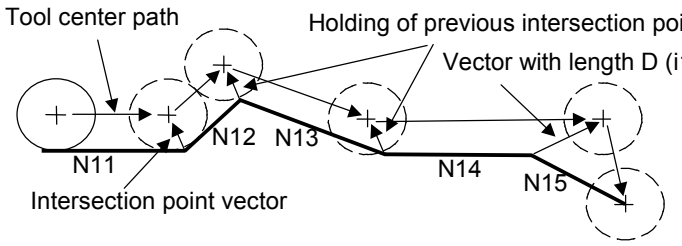
The compensation vector can be changed in following two ways.

G38 Xx1 Yy1 ;	
Xx1, Yy1	: Movement amount

The tool radius compensation vector amount and direction are retained.

G38 Xx1 Yy1 Ii1 Jj1 Dd1 ;	
Xx1, Yy1	: Movement amount
Ii1, Jj1	: Compensation vector direction
Dd1	: Compensation vector length

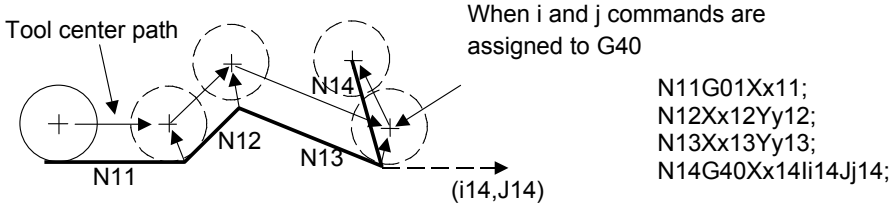
The tool radius compensation vector direction is updated by I and J.



The tool radius compensation is canceled by the following command.

G40 Xx1 Yy1 Ii1 Jj1 ;	
Xx1, Yy1	: Movement amount
Ii1, Jj1	: Compensation vector direction

The vector prior to canceling is prepared by calculating the intersection point with the I and J direction.

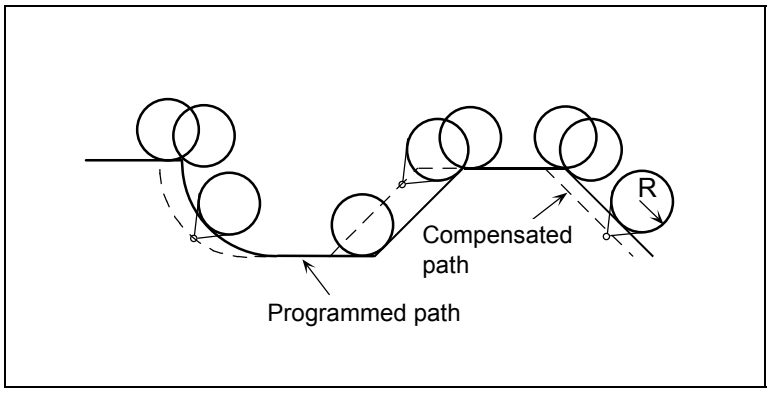


9.2.3 Tool Nose Radius Compensation (G40/G41/G42)

M system : - **L system : O**

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



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.

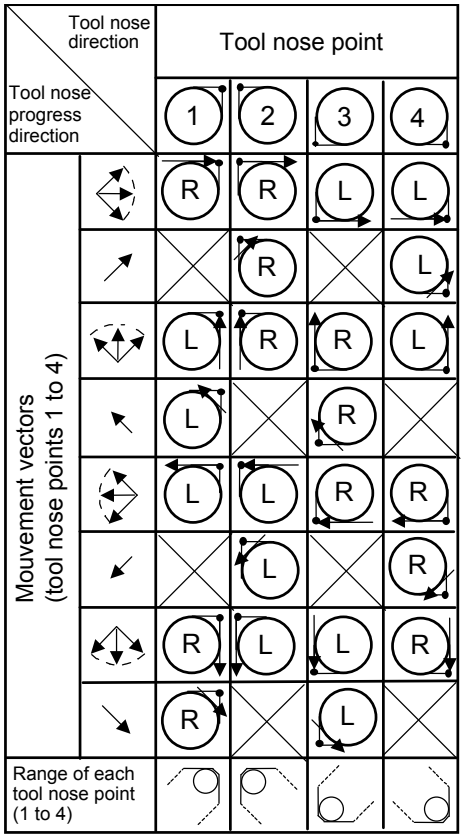
9.2.4 Automatic Decision of Nose Radius Compensation Direction (G46/G40)

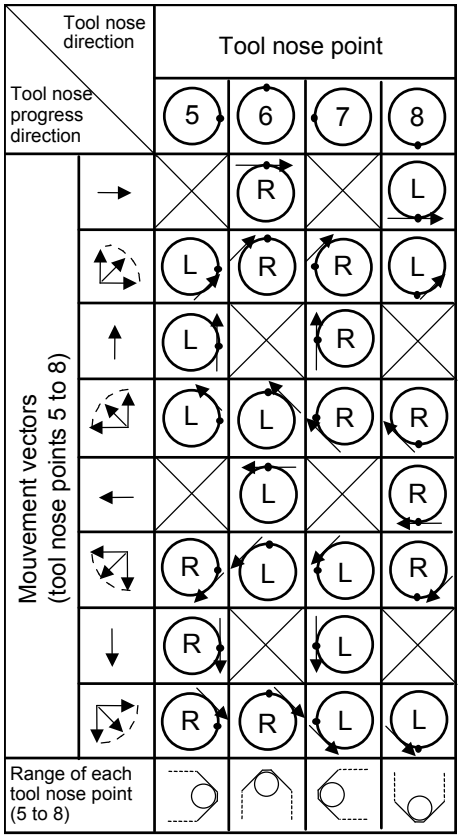
M system : - **L system : O**

The nose radius compensation direction is automatically determined from the tool tip 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 Offset Amount

9.3.1 Number of Tool Offset Sets

The number of tool compensation sets is as follows.

<M system>

Number of part systems / Number of tool compensation sets	1st part system	2nd part system	3rd part system	4th part system	5th part system	6th part system	7th part system
40 sets	○	○	○	○	○	○	○
80 sets	Δ	Δ	Δ	-	-	-	-
100 sets	Δ	Δ	-	-	-	-	-
200 sets	Δ	-	-	-	-	-	-

(Note 1) The number of tool compensation sets in above table indicates the number of sets in each part system.

(Note 2) The standard number of tool compensation sets per part system for M system is 40 regardless of number of part systems.

<L system>

Number of part systems / Number of tool compensation sets	1st part system	2nd part system	3rd part system
40 sets	-	-	-
80 sets	○	○	○
100 sets	-	-	-
200 sets	-	-	-

(Note 1) The number of tool compensation sets per part system for L system is 80 regardless of number of part systems.

9.3.1.2 40 sets

M system : ○ **L system : -**

9.3.1.3 80 sets

M system : Δ **L system : ○**

9.3.1.4 100 sets

M system : Δ **L system : -**

9.3.1.5 200 sets

M system : Δ **L system : -**

9.3.2 Offset Memory

9.3.2.1 Tool Shape/Wear Offset Amount

M system : O **L system : O**

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

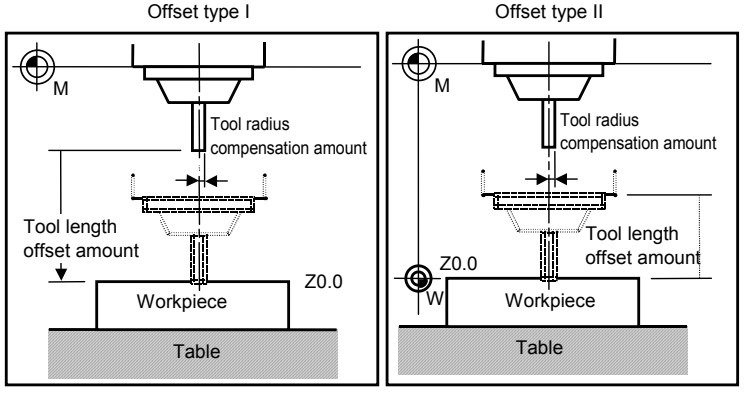
(1) Shape offset amount
The tool length offset amount, tool radius compensation amount, nose radius compensation amount, nose radius imaginary tool tip point or tool width can be set as the shape offset amount.
The compensation amount that can be set and used differs depending on whether offset amount setting type 1, 2 or 3 is used.

(2) Wear offset amount
When the tip of the tool used has become worn, the wear offset amount is used to offset this wear.
Types of wear offset amounts include the tool length wear offset amount, tool radius wear compensation amount, and nose radius wear compensation amount.
The wear offset amount can be used with offset amount setting types 2 and 3, and it is added to the shape offset amount for compensation.

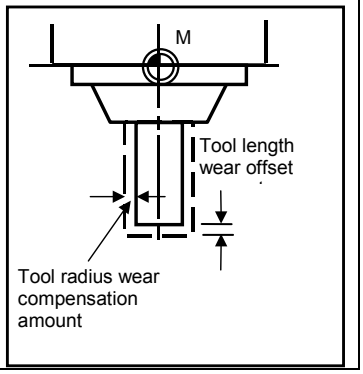
(a) Type 1: 1-axis offset amount [M system]
This is the value that is used by rotary tools.
As the tool length offset amount, among the offset amounts for the position of the tool moving in the direction parallel to the control axis, the offset amount in the longitudinal direction of the rotary tool is registered. The tool length offset amount is set as a minus value.
As the tool radius compensation amount, among the offset amounts for the position of the tool moving in the direction parallel to the control axis, the offset amount in the radial direction of the rotary tool is registered. The tool radius compensation amount is set as a plus value.
One offset amount data is registered in one offset number, and the offset Nos. are assigned using the address D or H commands. When a No. is assigned by a D address command, offset 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 2: 1-axis offset amounts/with wear offset [M system]
As with type 1, type 2 is for the offset amounts used by rotary tools.
With type 2, four kinds of offset amount data are registered in one offset No.: the tool length offset amount, tool length wear offset amount, tool radius compensation amount, and tool radius wear compensation amount.
When an offset No. is assigned by address D as the offset amount, the tool radius is compensated using the amount obtained by adding the tool radius compensation amount and tool radius wear compensation amount. Further, the tool length is offset using the amount obtained by adding the tool length offset amount and tool length wear offset amount.

Figure: Example of how the offset amount is handled when using the type 1 tool length offset amount (Offset types I and II are available for handling offset amounts.)



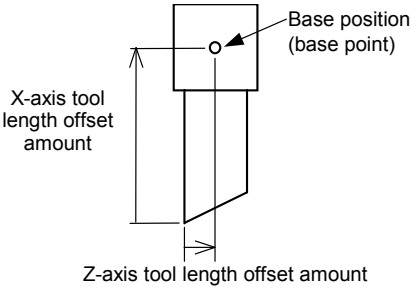
Wear offset amount when using type 2



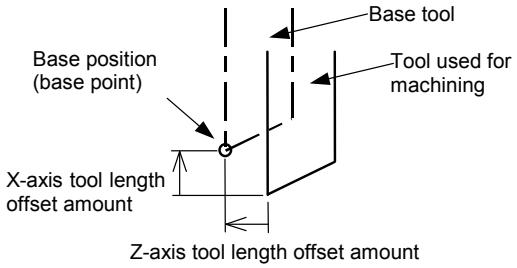
(c) Type 3: 2-axis offset amounts [L system]

Type 3 is for the offset amounts used by non-rotary tools.
As the offset amounts, the tool length along the X, Z axes and additional axis 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.
Offset is provided in the directions of the X, Z axes and additional axis from the base position in the program. Generally, the center of the tool rest or the tip of the base tool is used as the programmed base position.

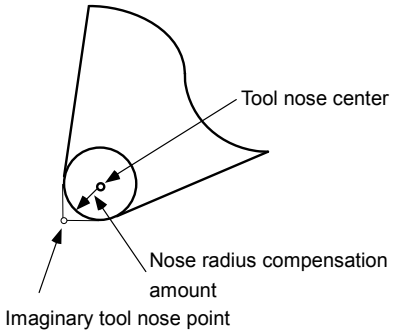
1. The programmed base position is the center of the tool rest:

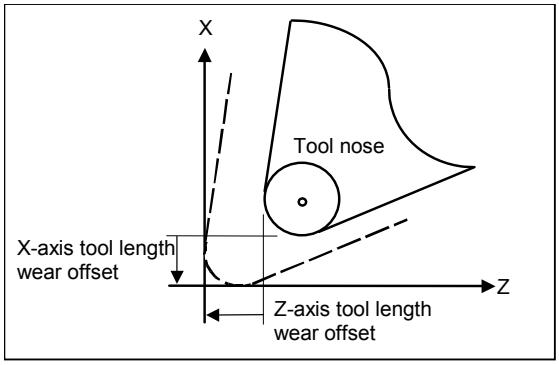


2. The programmed base position is the tip of the base tool:



The tool tip contour arc radius (nose radius) of a non-rotary tool with an arc (nose radius) at its tip is registered as the nose radius offset amount.



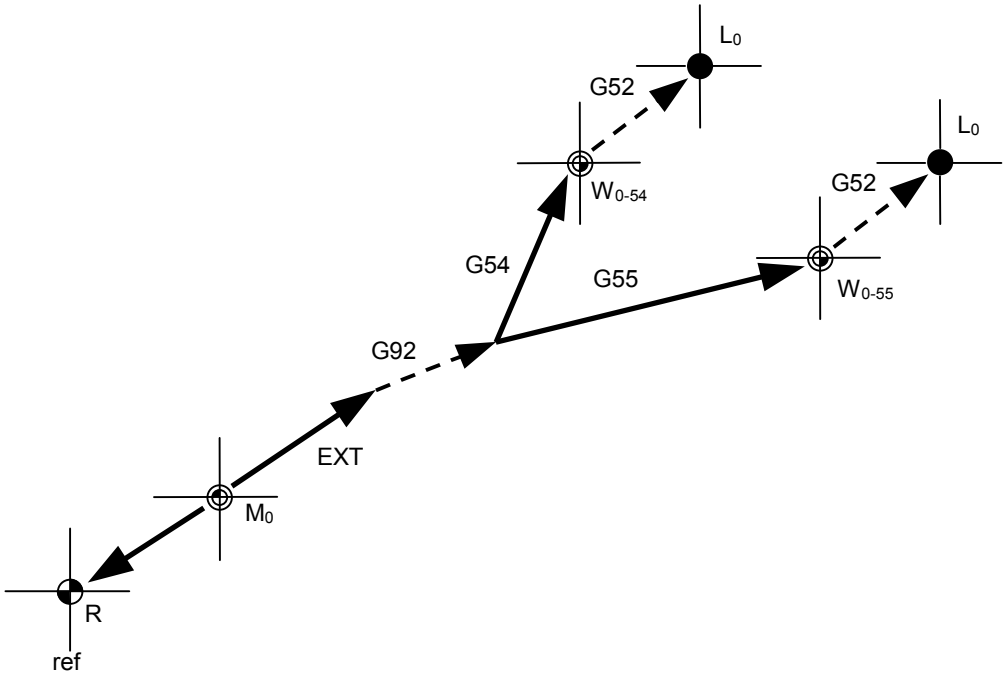


The X-axis tool length offset amount, Z-axis tool length offset amount and nose radius compensation amount are set as plus amounts.
The offset type (1, 2 or 3) is set using a parameter.

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.



L_0	Local coordinate system zero point	⟶	Offset set with parameters
G52	Local coordinate system offset *1)	---▶	Offset set with program
$W_{0\text{-}54}$	Workpiece coordinate system zero point (G54)		(0 when power is turned ON)
$W_{0\text{-}55}$	Workpiece coordinate system zero point (G55)		
G54	Workpiece coordinate system (G54) offset *1)		*1)The G52 offset is available
G55	Workpiece coordinate system (G55) offset		independently for G54 to G59.
G92	G92 coordinate system shift		
EXT	External workpiece coordinate offset		
M_0	Machine coordinate system zero point		
ref	Reference point		

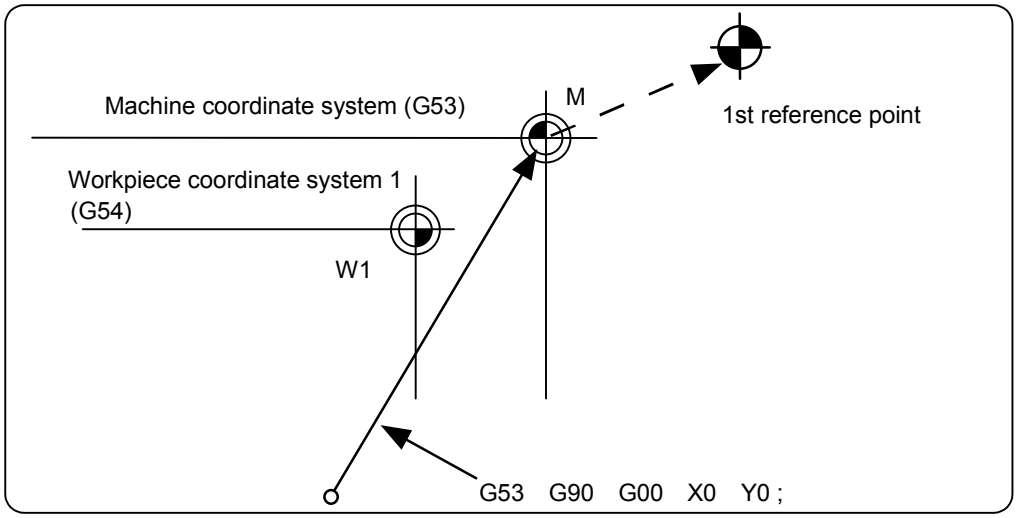
10.1.1 Machine Coordinate System

M system : O **L system : O**

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 point 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 to 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.

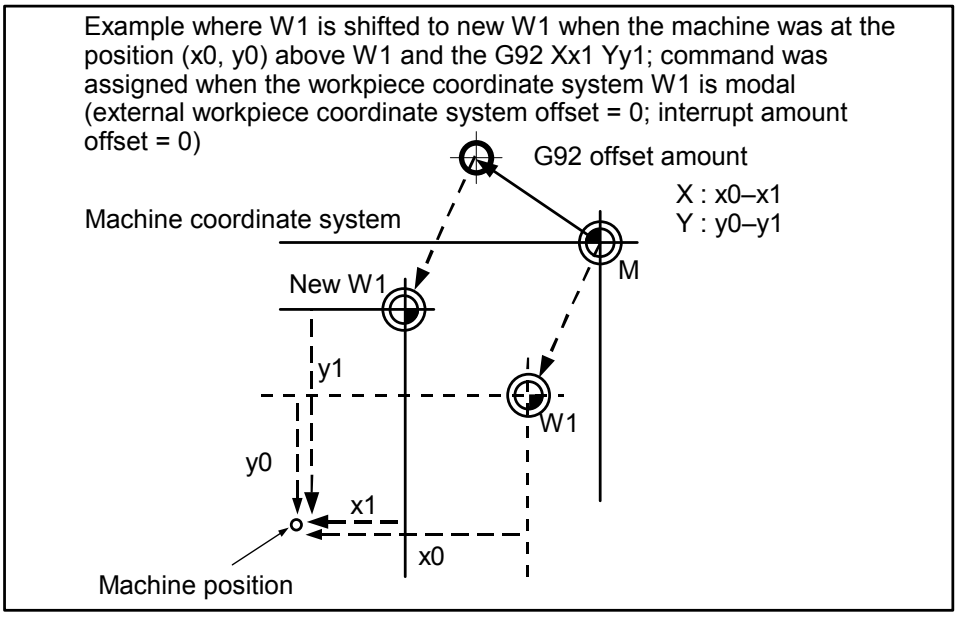


10.1.2 Coordinate System Setting

M system : O **L system : O**

By giving a G92 command, the program coordinate system (zero point of program) can be changed on the workpiece coordinate system.
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 values 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

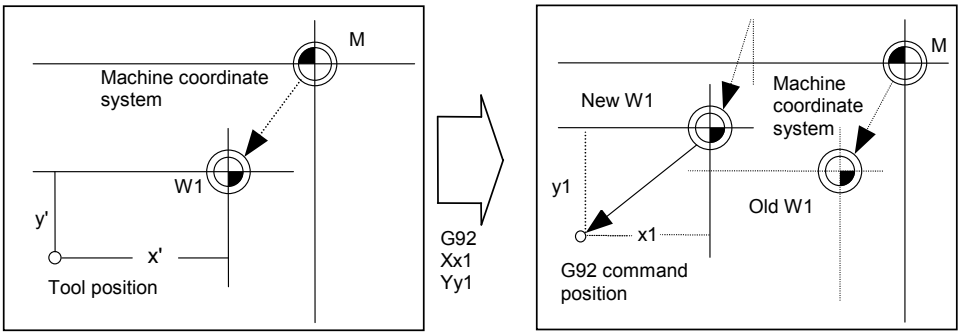


The shifted coordinate system is returned to its original position by dog-type reference point return or the program.

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.

Coordinate system created by automatic coordinate system setting

Coordinate system after coordinate system setting by G92



(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 point 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 ; G92 G53 X0 Y0 ;	Positioning at machine coordinate system zero point. 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 : O **L system : O**

After the power is turned ON, the basic machine coordinate system and the workpiece coordinate system are automatically set without executing the zero point return.
The coordinate systems created are given below.

(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. Thus, where the No. 1 reference point is set in the machine is the base for the setting.

10.1.4 Workpiece Coordinate System Selection

10.1.4.1 Workpiece coordinate system selection (6 sets) G54 to G59

M system : ○ **L system : ○**

When multiple 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.

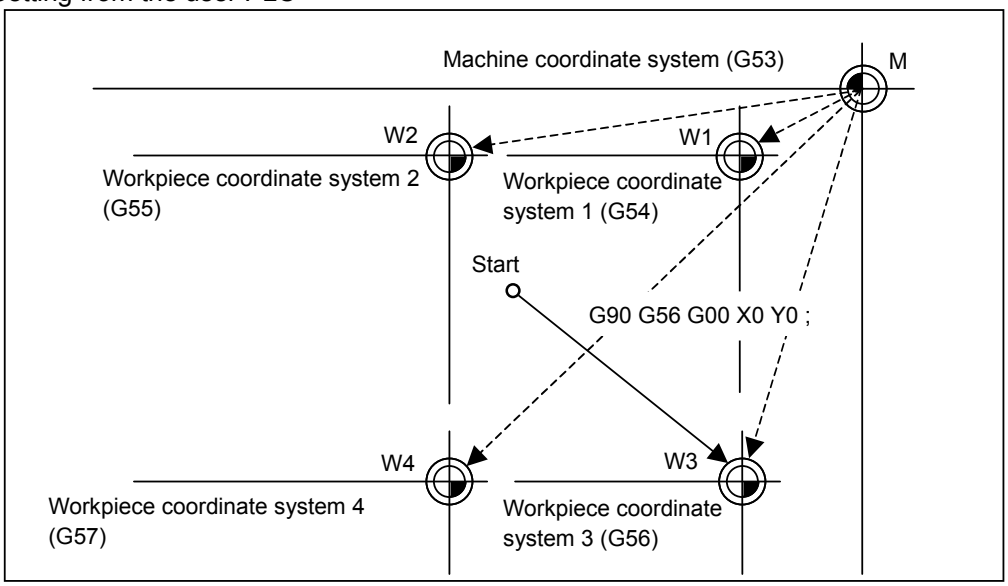
G code	Function
G54	Workpiece coordinate system 1 (W1)
G55	Workpiece coordinate system 2 (W2)
G56	Workpiece coordinate system 3 (W3)
G57	Workpiece coordinate system 4 (W4)
G58	Workpiece coordinate system 5 (W5)
G59	Workpiece coordinate system 6 (W6)

The command format 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:

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



10.1.4.2 Extended workpiece coordinate system selection (48 sets) G54.1P1 to P48

M system : Δ **L system : -**

When multiple 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:

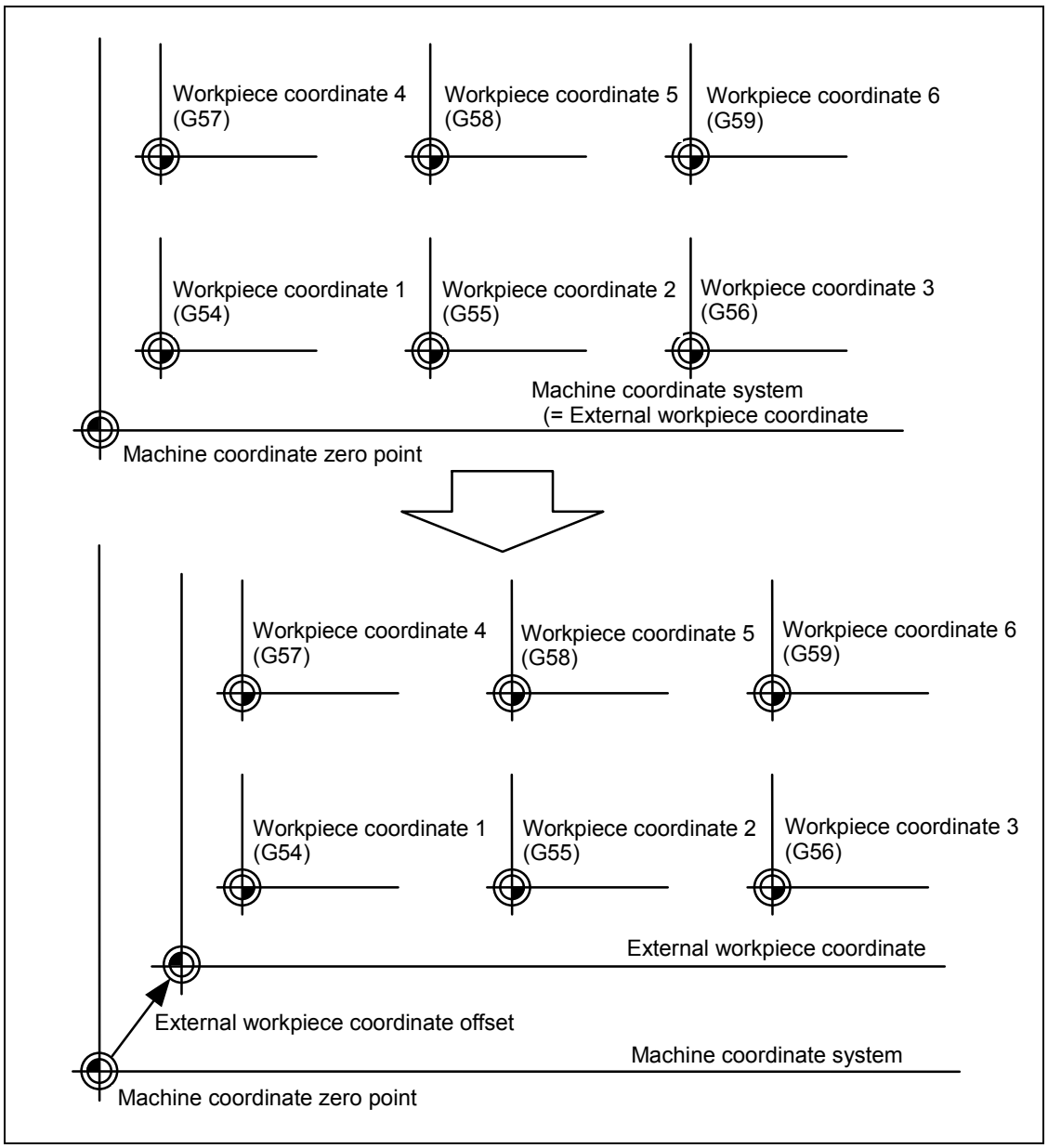
(a) Setting using the setting and display unit
(b) Setting using commands assigned from the machining program
(c) Setting from the user PLC

(Note 1) 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 : O **L system : O**

An external workpiece coordinate offset that serves as a 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 by selecting the external workpiece coordinates.



10.1.7 Local Coordinate System

M system : O **L system : O**

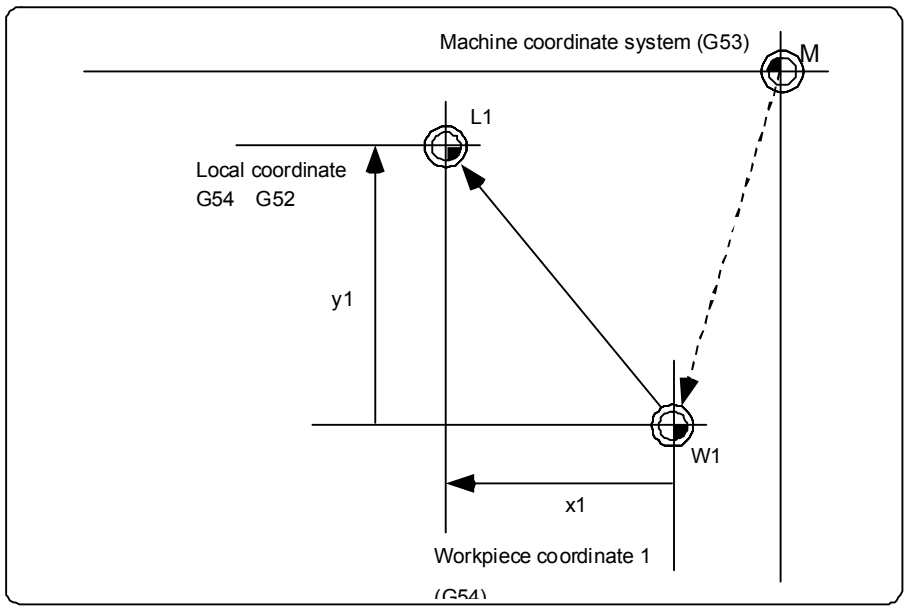
This function is for assigning a coordinate system on the workpiece coordinate system currently being selected. This enables the workpiece coordinate system to be changed temporarily.
The local coordinate system can be selected independently on each workpiece coordinate system G54 to G59.

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 value 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 point return.



10.1.8 Coordinate System for Rotary Axis

M system : O **L system : O**

The coordinate system of the rotary axis ranges from 0 to ±360°. Note that, however, it can be displayed from 0 to 359.999.
In absolute value command mode, the rotary axis can make a turn or less (not greater than ±360°). The turning direction depends on the specified sign. A negative sign (–) turns the axis in the negative direction and a positive sign (+) turns it in the positive (+) direction.
Note that a parameter can be used to move the axis to the end point taking a short cut.
In incremental value command mode, the rotary axis moves the specified distance only.

10.1.9 Plane Selection

M system : O **L system : O**

G17, G18, and G19 are for specifying the planes for the arc, tool radius compensation, coordinate rotation and other 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.

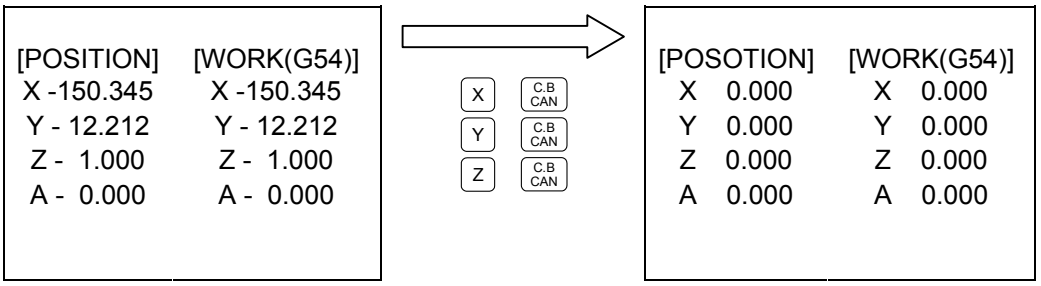
Example

G19 X100. ;	With these program commands, X100. is the axis which does not exist on the G19 (Yp, Zp) plane, Yp-Zp are 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 : O **L system : O**

Using the CNC monitor, the coordinate system (current position and workpiece coordinate position) can be set to "0" by screen operations. This function is the same as the coordinate system setting command "G92 X0 (Y0 or Z0) ; ".



When axes are set to "0" in order, the Y and Z axis can be set by pressing [C.B CAN] key successively without pressing [Y] and [Z] keys.

10.1.11 Counter Set

M system : O **L system : O**

Using CNC monitor, the position counter display can be changed to "0" by screen operations.

(1) This operation is the same as the operation of "Origin Set", but press [INPUT] key instead of [C.B CAN] key.

(2) Only the [POSITION] counter display is changed to "0", and the other coordinate system counter displays are not changed.

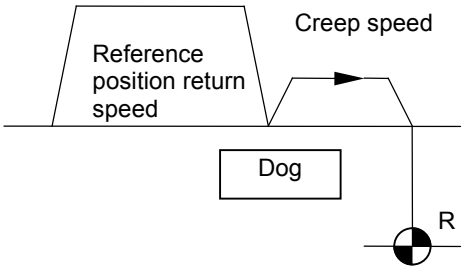
10.2 Return

10.2.1 Manual Reference Position Return

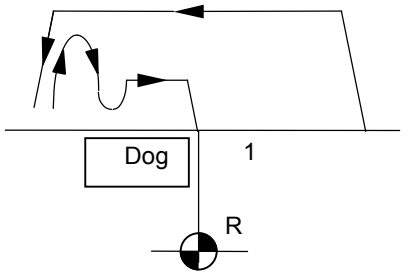
M system : O **L system : O**

This function enables the tool to be returned manually to the machine's default position (reference position).
(1) Return pattern to reference point

(a) Dog type

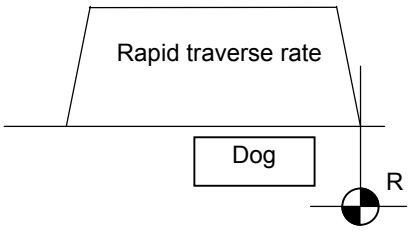


When starting in same direction as final advance direction



When starting in opposite direction as final advance direction

(b) High-speed type



(2) Differences according to detection method

	First return after power ON	Second return and following
Incremental position detection method	Dog-type	High-speed/Dog-type (switching by parameter)
Absolute position detection method	High-speed	High-speed

10.2.2 Automatic 1st Reference Position Return

M system : O **L system : O**

The machine can be returned to the first reference point 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 point.
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 command position.

G code	Function
G28	Automatic 1st reference point return
G29	Start position return (The tool first returns to the interim position of the 1st reference point return start from the 1st reference point, and then is positioned at the position designated in the program.)

The G28 programming format is given below.

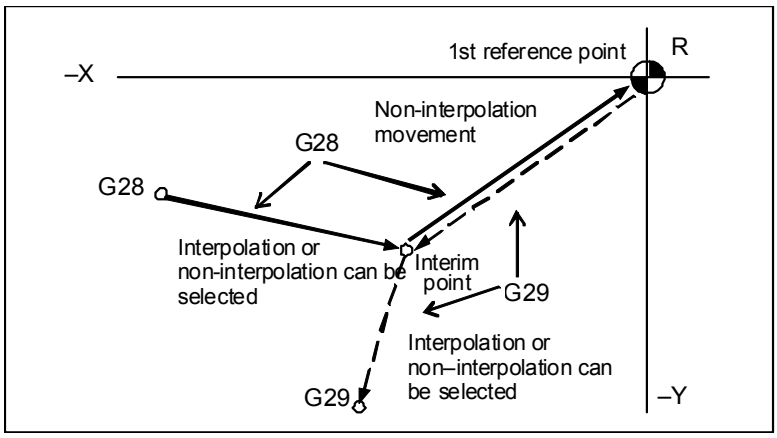
G28 Xx1 Yy1 Zz1 ;
G28 : Return command
Xx1, Yy1, Zz1 : Return control axes (interim point)

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

The G29 programming format is given below.

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

The tool is first moved by rapid traverse 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.



If the position detector is for the incremental detection system, the first reference point return for the first time after the NC power has been turned ON will be the dog-type. However, the second and subsequent returns are to be the high-speed type.
The high-speed type is always used when the position detector is for the absolute position detection system.

(Note 1) The automatic 1st reference point return pattern is the same as for manual reference point return.
(Note 2) The number of axes for which reference point return can be performed simultaneously depends on the number of simultaneously controlled axes.
(Note 3) If, at the time of the first reference point return, the tool radius compensation or nose radius compensation has not been canceled, it will be temporarily canceled by the movement to the interim point. The compensation is restored by the next movement after the return.
(Note 4) If, at the time of the first reference point return, the tool length offset has not been canceled, the offset will be canceled by the movement from the interim point to the first reference point, and the offset amount will also be cleared. It is possible to cancel the tool length offset temporarily using a parameter instead. In this case, however, the offset 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 point and movement up to the G29 interim point.
(Note 6) The machine will not stop at the interim point even when a single block is selected.

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

M system : O **L system : O**

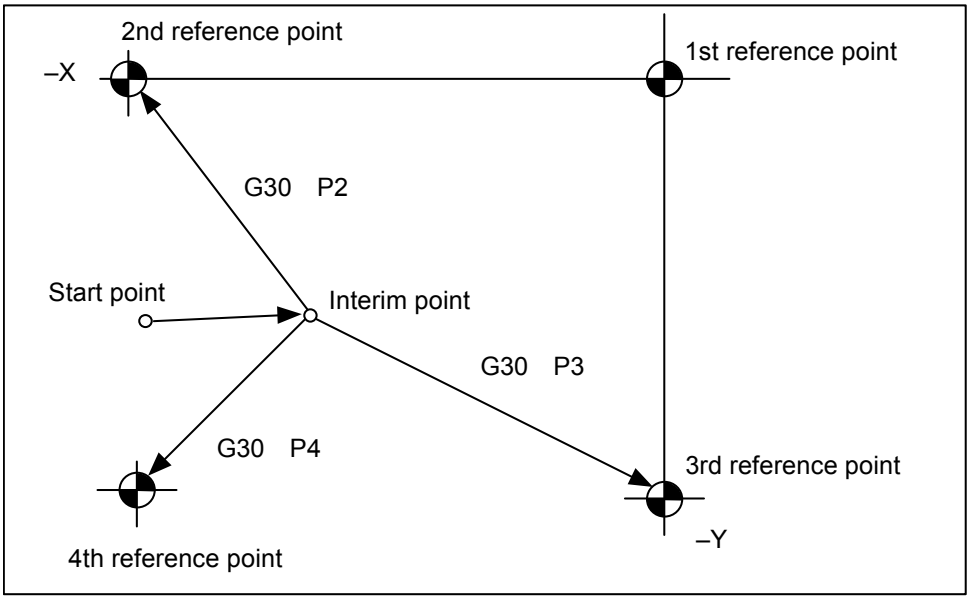
As with automatic 1st reference point return, commanding G30Pn during automatic operation enables the tool to be returned to the set points (2nd, 3rd or 4th reference points) characteristic to the machine. The 2nd, 3rd and 4th reference points can be set by parameters.

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

The G30 programming format is given below.

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

The tool is first positioned by rapid traverse to the interim point commanded for the assigned axis and then is returned independently to the reference point.



(Note 1) The second reference point return is performed if the P address is omitted.
(Note 2) The number of axes for which reference point return can be performed simultaneously depends on the number of simultaneously controlled axes.
(Note 3) If, at the time of the reference point return, the tool radius compensation has not been canceled, it will be temporarily canceled by the movement up to the interim point. The compensation is restored by the next movement command after the return.
(Note 4) If, at the time of the reference point return, the tool length offset has not been canceled, it will be canceled and the offset amount also cleared upon completion of reference point return. The tool length offset can also be canceled temporarily using a parameter. In this case, however, the tool offset 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 points.
(Note 6) The machine will not stop at the interim point even when a single block is selected.

10.2.4 Reference Position Check

M system : O **L system : O**

By commanding G27, a machining program, which has been prepared so that the tool starts off from the reference point and returns to the reference point, can be checked to see whether the tool will return properly to the reference point.
The G27 programming format is given below.

G27 Xx1 Yy1 Zz1 Pp1 ;	
G27	: Verification command
Xx1, Yy1, Zz1	: Return control axes
Pp1	: Verification No.
	P1 : 1st reference point verification
	P2 : 2nd reference point verification
	P3 : 3rd reference point verification
	P4 : 4th reference point verification

The assigned axis is first positioned by rapid traverse to the commanded position and then, if this is the reference point, the reference point arrival signal is output.
When the address P is omitted, the first reference point verification will be applied.

(Note 1) The number of axes for which reference point verification 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 point 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 : Δ **L system : Δ**

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 point return. (High-speed return will always be used for the reference point return command.)
For the absolute position detection method, there are two method 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-less type	Machine end stopper method	Manual	The zero point is established by manually 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 absolute position setting screen.
		Automatic	The zero point is established by automatically 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 execute an automatic pressing twice.	
	Marked point alignment method	Method I	The zero point is established by aligning with a marked point on the machine. It is established after aligning with a marked point and then returning to the grid point.	The zero point is established by input in the absolute position setting screen.	The value equivalent to the shift amount is set in the absolute position setting screen.
		Method II	The zero point is established by aligning with a marked point on the machine. It is established after aligning with a marked point and but not returning to the grid point.		
Dog-type			Same method as the dog-type of incremental detection method.	The zero point is established with dog-type reference point return completion.	The value is set in the parameter of zero point shift amount.

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.

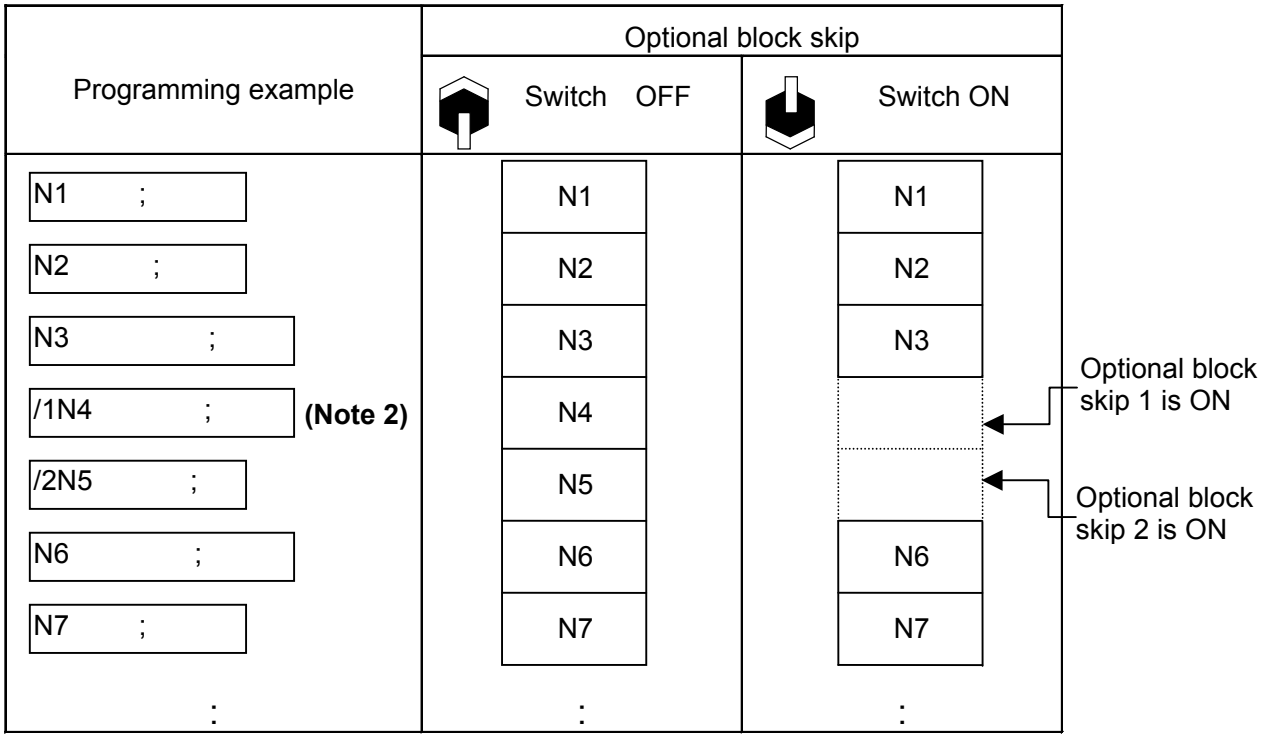
11. Operation Support Functions

11.1 Program Control

11.1.1 Optional Block Skip

M system : O **L system : O**

When "/" (slash code) is programmed at the head of a block, and 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.



(Note 1) There are nine optional block skip switches corresponding to "/".
(Note 2) "1" of "/1N4" can be omitted.

11.1.2 Optional Block Skip Addition

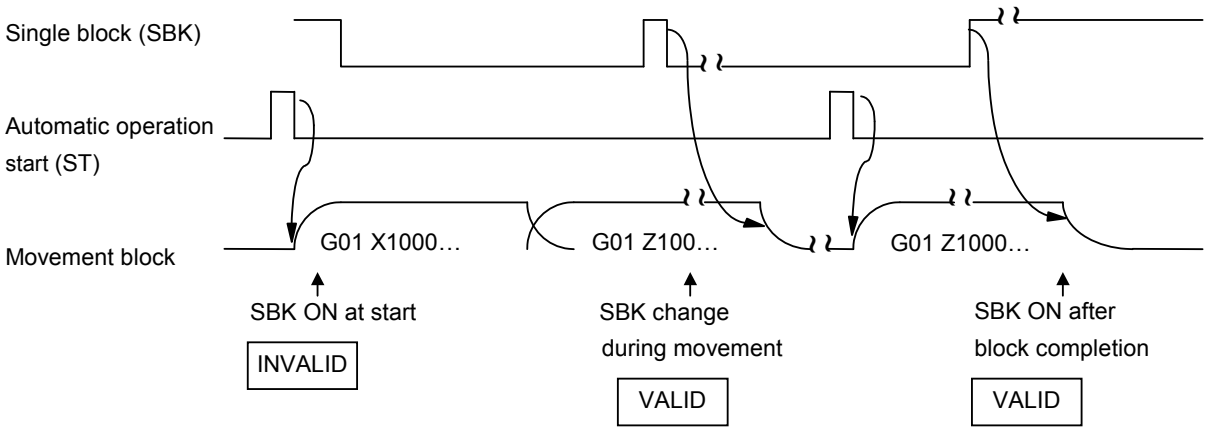
M system : O **L system : O**

When "/n (n:1 to 9)" (slash code) is programmed at the head of a block, and the optional block skip n input signal from the external source is turned ON for automatic operation, the block with the "/n" code is skipped.

11.1.3 Single Block

M system : O **L system : O**

The commands for automatic operation can be executed one block at a time (block stop) by turning ON the single block input signal. When the single block input signal is turned ON temporarily during continuous operation, the machine will stop after that block has been executed.
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.
Single block in the multi-part system also functions as the above single block in each independent part system.



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 selector switch OFF	Rapid traverse selector switch ON
G00, G27, G28, G29, G30, G60	Manual feed rate **(Note 1)**	Rapid traverse rate
G01, G02, G03	Manual feed rate	Cutting clamp speed

(Note 1) The dry run should be valid by the parameter setting.

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 actually moving the NC axis.
The command speed is the feed rate during machine lock.
Cutting override and rapid traverse override are valid.
The M, S, T and B commands are executed as usual, and so machine lock is completed by returning the FIN signal.

(1) Reference point return (manual, G28, G29, G30) is controlled as far as the interim point in the machine lock status but when the interim point is reached the counter is moved to the zero point and the block is completed.
(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 and OFF or OFF and ON during automatic operation.
(4) On PLC programming, the signal for machine lock has for automatic operation and manual operation of each axis. Normally, all signals are simultaneously turned ON and OFF. However when Z axis cancellation function is executed, the machine lock signal for Z axis is turned ON and OFF.

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.

11.3 Program Search/Start/Stop

11.3.1 Program Search

M system : O **L system : O**

The program No. of the program to be operated automatically can be designated and called up. 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

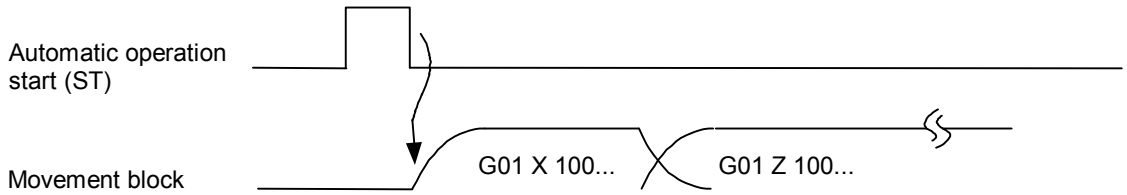
M system : O **L system : O**

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.

11.3.5 Automatic Operation Start

M system : O **L system : O**

With the input of the automatic operation start signal (change from ON to OFF), automatic operation of the program that was found by an operation search is started by the controller (or the halted program is restarted).



Automatic operation startup is performed on a part system by part system basis.

11.3.6 NC Reset

M system : O **L system : O**

This function enables the controller to be reset.

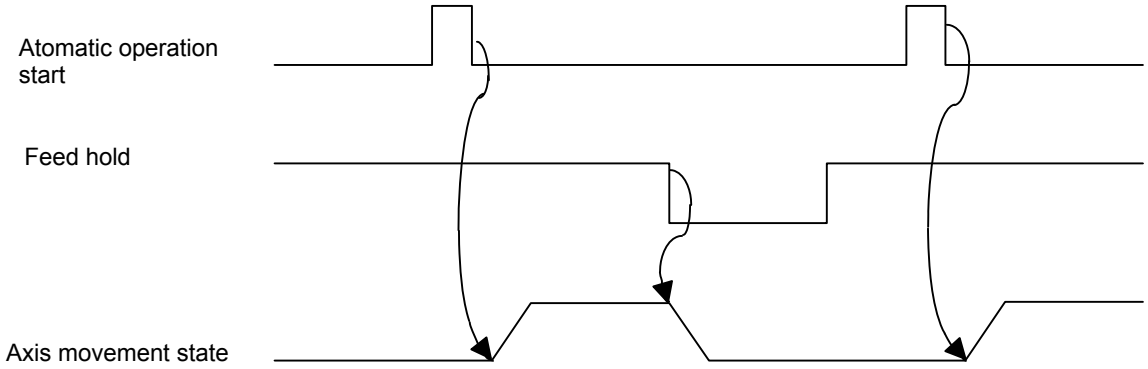
	Signal name / Target	Reset 1	Reset 2	Reset & Rewind
1	G command modals	Retained	Initialized	Initialized
2	Tool compensation data	Retained	Canceled (no operations)	Canceled
3	Memory indexing	Executed	Not executed	Executed
4	Errors/alarms	Reset	Reset	Reset
5	M, S and T code outputs	Retained	Retained	Retained
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 : O **L system : O**

When the feed hold signal is set to 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 (jog feed, handle feed or incremental feed), the feed hold stop mode is entered.
(3) An interrupt operation based on manual operation (jog feed, handle feed or incremental feed) can be executed during feed hold.



11.3.8 Search & Start

M system : O **L system : O**

If the "Search & Start" signal is input when the memory mode is selected, the designated machining program is searched and executed from the beginning.
If the search & start signal has been input during automatic operation in the memory mode, search & start is executed after resetting.
The machining program No. to be searched are designated by PLC program.

11.4 Interrupt Operation

11.4.1 Manual Interruption

M system : O **L system : O**

Manual interrupt is a function that enables manual operations to be performed during automatic operation. The systems used to select the operation mode are as follows:

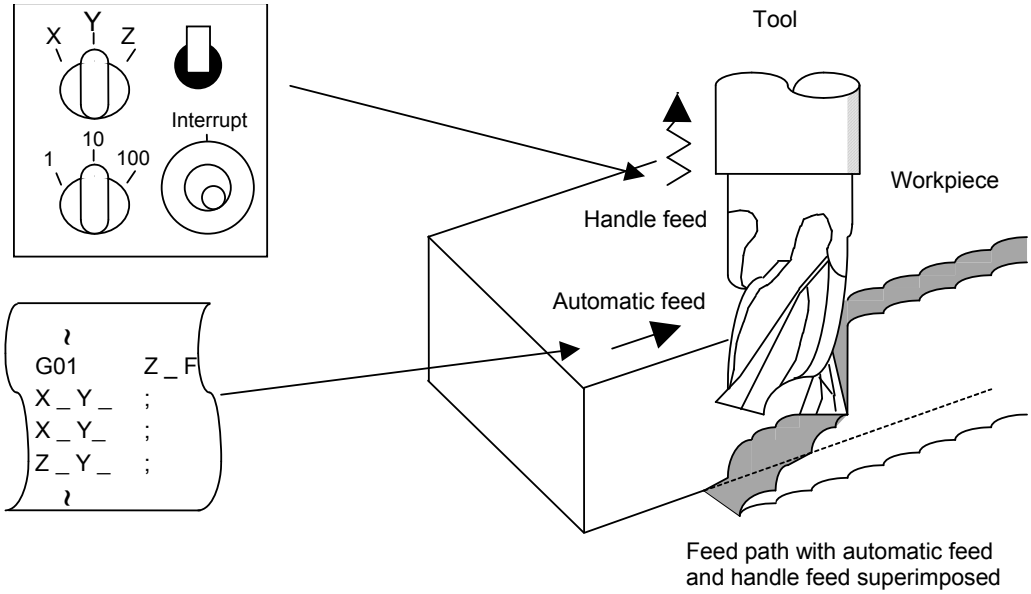
- System which initiates the interrupt by switching from the automatic mode to manual mode
- System which initiates the interrupt by selecting the manual mode at the same time as the automatic mode
  (Refer to "11.4.9 Simultaneous Operation of Manual and Automatic Modes".)

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 Mode ON/OFF").

11.4.2 Automatic Operation Handle Interruption

M system : O **L system : O**

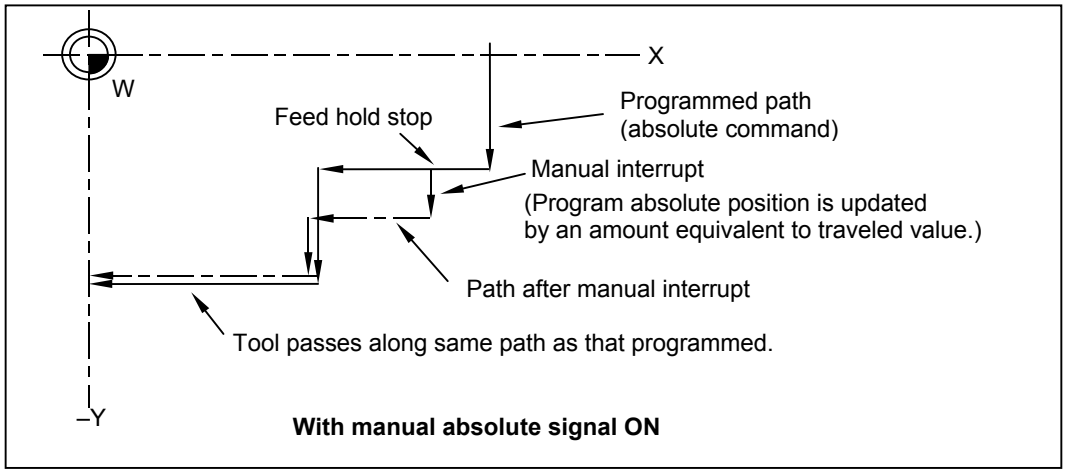
The handle command can interrupt and be superimposed onto a command without suspending automatic operation, and the machine can be moved by rotating the manual pulse generator during automatic operation.
If the spindle load is greatly exceeded when cutting a workpiece as per the machining program due to a high rough cutting amount in face machining, for instance, automatic handle interrupt makes it possible to raise the Z surface and reduce the load easily without suspending feed in the automatic operation mode.
Automatic handle interrupt is conducted by setting the "automatic handle interrupt" valid switch which is provided separately from the "manual operation mode". The axis selection and pulse scale factor 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.



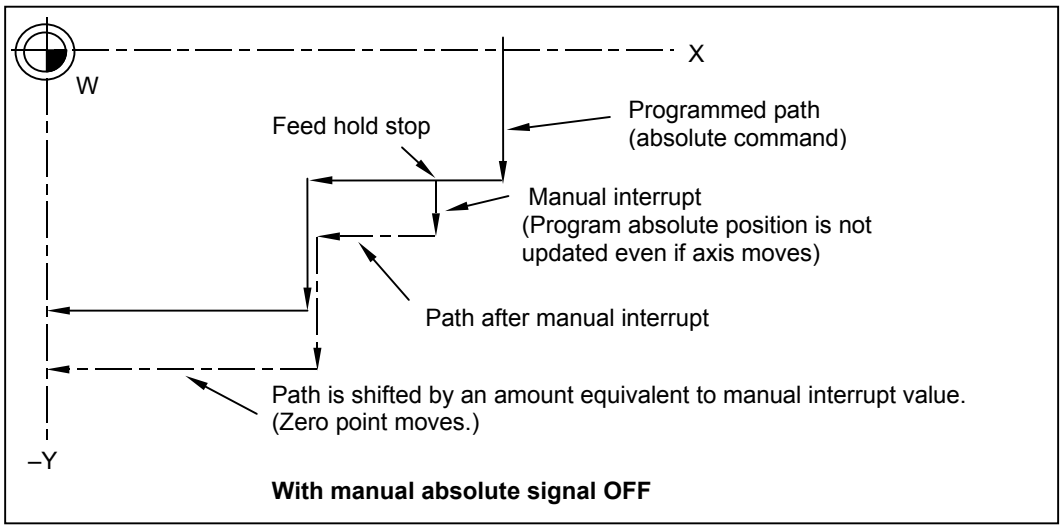
11.4.3 Manual Absolute Switch

M system : O **L system : O**

The program absolute positions are updated by an amount equivalent to the distance by which the tool is moved by hand when the manual absolute switch signal is turned ON.
In other words, the coordinate system based on the original program will not shift even if the tool (machine) is moved by hand. Thus, if automatic operation is started in this case, the tool will return to the path before manual movement.



With manual absolute signal ON

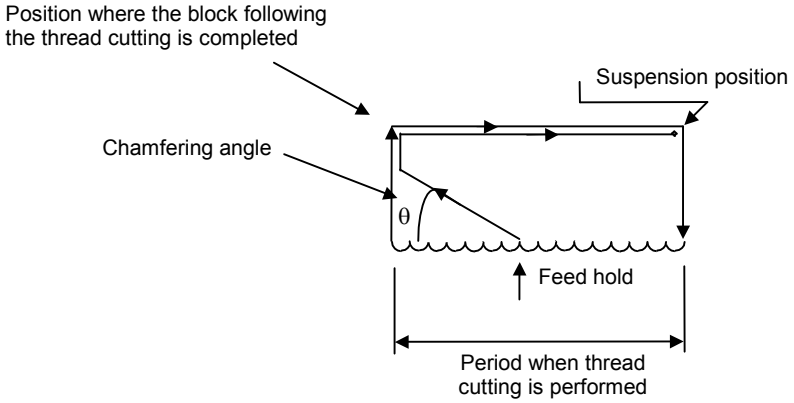


With manual absolute signal OFF

11.4.4 Thread Cutting Cycle Retract

M system : - **L system : Δ**

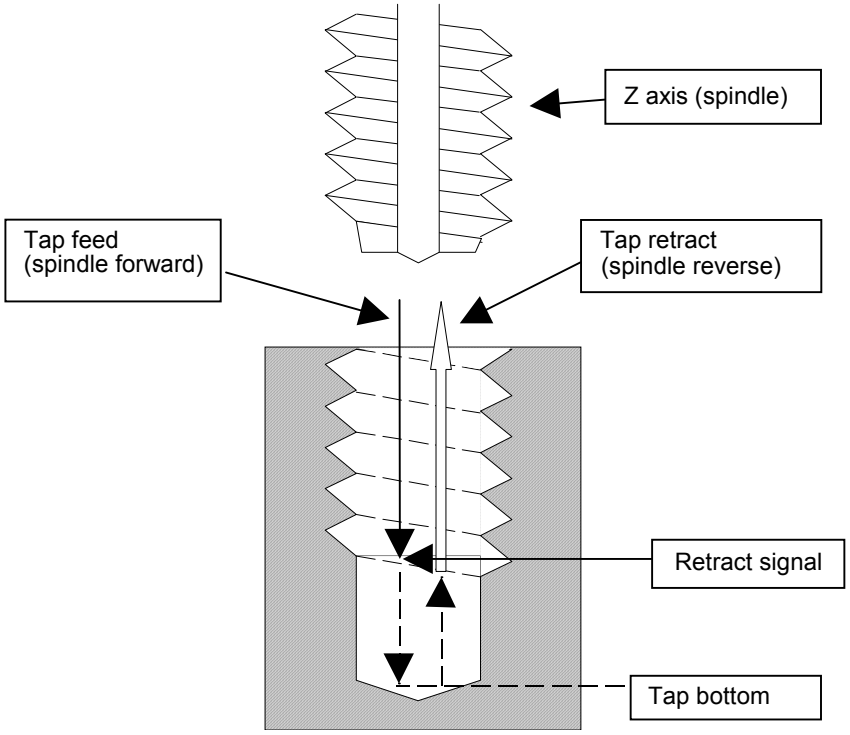
This function suspends the thread cutting cycle if a feed hold signal has been input during thread cutting cycle.
If a feed hold signal is input during chamfering or thread cutting without chamfering, operation stops at the position where the block following the thread cutting is completed.



11.4.5 Tapping Retract

M system : O **L system : O**

If tapping is interrupted by a reset or emergency stop signal that is input during tapping and the tap is left engaged inside the workpiece, the tap tool engaged inside the workpiece can be rotated in the reverse direction so that it will be disengaged by inputting the tap retract signal.

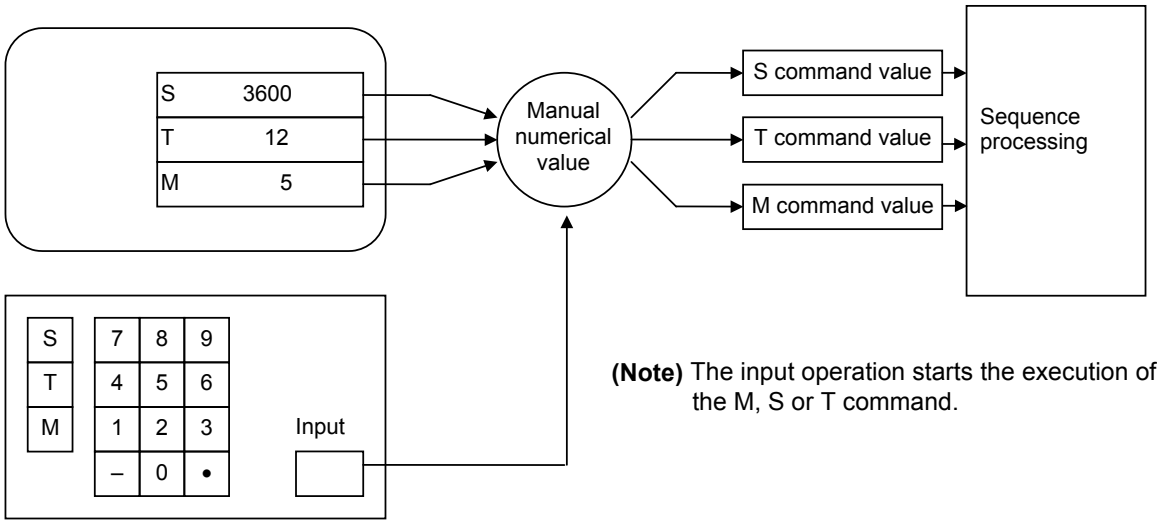


This function can be used by an interruption initiated by reset or emergency stop.
A return is made to the initial point by tap retract.

11.4.6 Manual Numerical Value Command

M system : O **L system : O**

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 offset 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.



(Note) The input operation starts the execution of the M, S or T command.

11.4.8 MDI Interruption

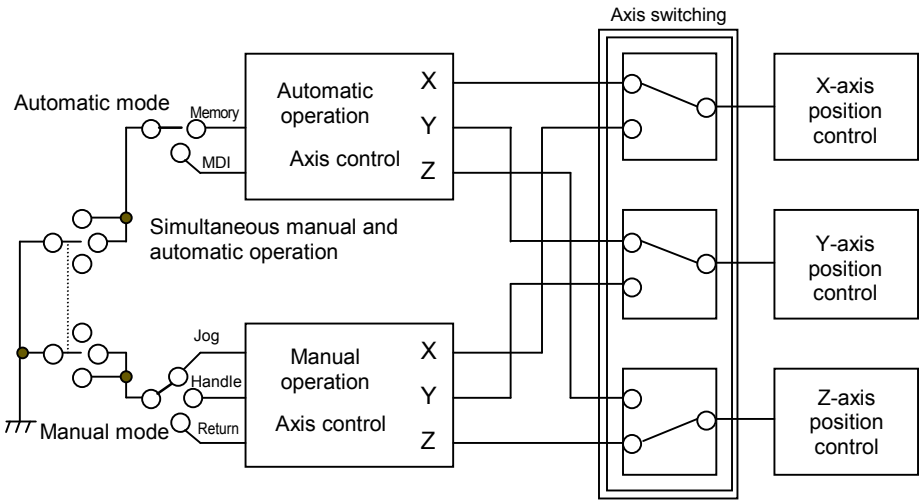
M system : O **L system : O**

This function enables MDI programs to be executed during automatic operation in the single block stop status. When the modal status is changed in the MDI program, the modal status in the automatic operation mode is also changed.

11.4.9 Simultaneous Operation of Manual and Automatic Modes

M system : O **L system : O**

This function enables manual operations to be performed during automatic operation by selecting an automatic operation mode (MDI or memory) and manual mode (handle, step, jog or manual reference point 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 axes subject to 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 applies to axes subject to automatic commands.

12. Program Support Functions

12.1 Machining Method Support Functions

12.1.1 Program

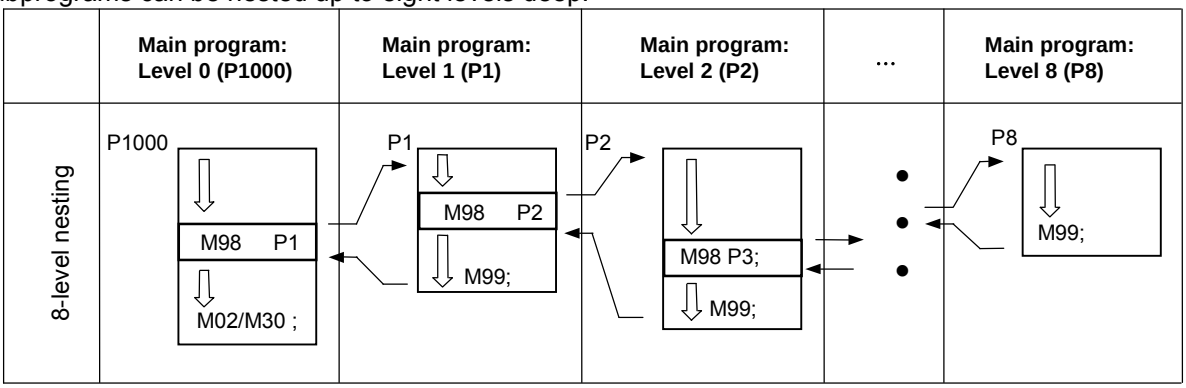
12.1.1.1 Subprogram Control

M system : O 8 layers L system : O 8 layers

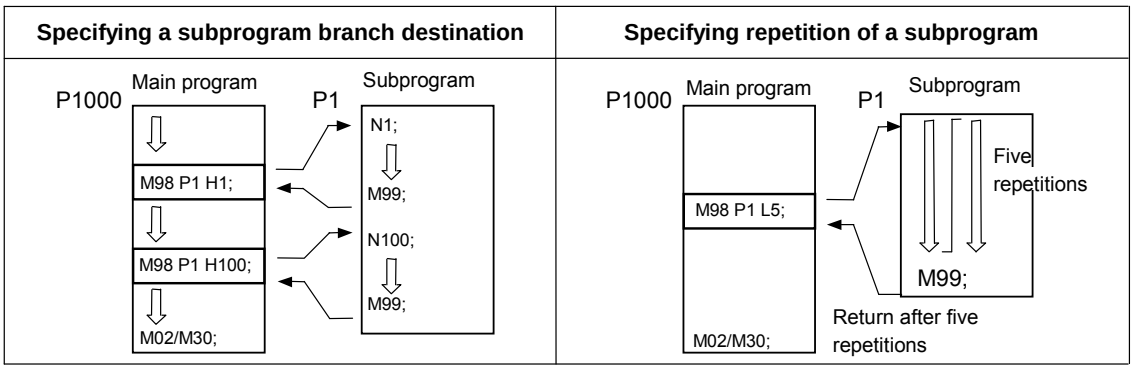
When the same pattern is repeated during machining, the machining pattern is registered as one subprogram and the subprogram is called from the main program as required, thereby realizing the same machining easily. Efficient use of program is possible. The call is designated with the program number and sequence number.

M98 Pp1 Hh1 Ll1 ;
M98 : Call command
Pp1 : Subprogram number
Hh1 : Sequence number
Ll1 : Number of repetitions
(Branch to subprogram)
Op1 (Subprogram)
:
Nh1
:
M99 ; (Return to main program)

Subprograms can be nested up to eight levels deep.



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



12.1.2 Macro Program

12.1.2.1 User Macro

M system : Δ 4 layers L system : Δ 4 layers

(1) Macro commands (1) ; G65 to G67
In order to execute 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 Ll1 Argument ;
G65 : Call command
Pp1 : Program No.
Ll1 : No. of repetitions
Argument : Variable data assignment

The macro program is called immediately by this command.

G66 Pp1 Ll1 Argument ;
G66 : Call command
Pp1 : Program No.
Ll1 : No. of repetitions
Argument : Variable data assignment

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

G66.1 Pp1 Ll1 Argument ;
G66.1 : Call command
Pp1 : Program No.
Ll1 : No. of repetitions
Argument : 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.

(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

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 G00 to G255 can be used for this command. (Whether to use codes such as G00, G01 or G02 which have already been clearly assigned for specific applications by the EIA standards as macro codes can be changed over using a parameter.)

(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

Mm ; (or Ss;, Tt;, Bb;)
Mm (Ss, Tt, Bb) : M (or S, T, B) code for performing macro call

The correspondence between the Mm 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 M95 can be entered. Select codes to be entered which are not the codes basically required by the machine and which are not M codes M0, M1, M2, M30 and M96 through M99.

(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.)

(Note 2) The registration of the program number used for calling the G code macro or M code macro can be done independently for each system. [M system]

12.1.2.3 Macro Interruption

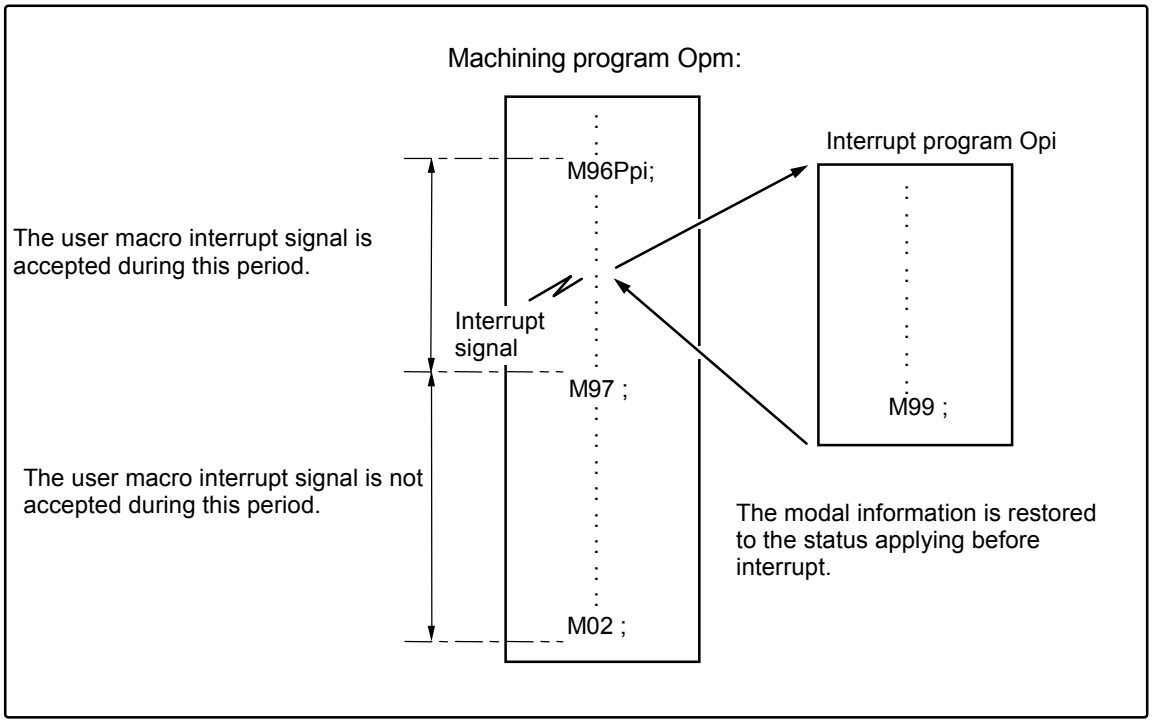
M system : Δ **L system : Δ**

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 P__ H__ ;	**User macro interrupt valid**
M97 ;	**User macro interrupt invalid**
P	: Interrupt program No.
H	: Interrupt sequence No.



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
The numbers of common variable sets depend on the options, and are as follows.

Type		Number (inside the brackets indicate No. of variable sets)		Function
Common variables		Common variables 1	Common variables 2	Can be used commonly for main, sub and each macro program. S: Number of part systems
For 1-part system specifications	100 sets (A)	500 to 549 (50)	100 to 149 (50)	
	200 sets (B)	500 to 599 (100)	100 to 199 (100)	
	300 sets (C)	500 to 699 (200)	100 to 199 (100)	
	600 sets (D)	500 to 999 (500)	100 to 199 (100)	
For multi-part system specifications	100 sets (A)	500 to 549 (50)	100 to 149 (50) × S	
	200 sets (B)	500 to 599 (100)	100 to 199 (100) × S	
	300 sets (C)	500 to 699 (200)	100 to 199 (100) × S	
	600 sets (D)	500 to 999 (500)	100 to 199 (100) × S	
Local variables		1 to 33 (33)		Can be used as local variable in macro program.
System variables		1000 to 1395		Macro interface input/output
		2000 to 2800 10000 to 18000		Read and write of tool compensation data.
		3000		Macro alarm message
		3001, 3002		Integrated time
		3003		Changing the validity (valid or invalid) of single block/miscellaneous function.
		3004		Changing the validity (valid or invalid) of feed hold.
		3006		Message display and block stop
		3007		Changing the validity (valid or invalid) of mirror image.
		3011, 3012		Reading and writing of current date and time.
		3901, 3902		Reading and writing of the number of the workpiece machining and the maximum number of workpiece machining.
		4001 to 4021, 4101 to 4130 4201 to 4221, 4301 to 4330		Reading of G command modal and other modal information.
		5001 to 5141		Reading information of various positions.

Type	Number (inside the brackets indicate No. of variable sets)	Function
(cont.)	5201 to 532n	Reading and writing of workpiece coordinate system offset data.
	30060 to 30068	Reading the coordinate rotation parameter.
	31001 to 31023	Reading and writing of a rotation axis configuration parameter.
	31100, 31101	Reading the number of available blocks for reverse run and the counter of available blocks for reverse run.
	50000 to 51199	Reading and writing of the data between NC machining program and PLC program.
	60000 to 64700	Reading and writing of the tool life management data,
Fixed cycle variables	1 to 32 (32)	Local variables in a 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) Common variables can be classified into the following two types.
Common variable 1: Variables that can be commonly used throughout all the part systems.
Common variable 2: Variables that can be used in the program of the target part system.
(Note 4) Variable names can be set for #500 to #519.
(Note 5) Re-format is not required even after changing the option parameter of the number of variable sets. After changing the option parameter, the changed number of sets can be used by recycling the power.
(Note 6) System variables 50000 to 51199 are held even when the power is turned OFF.
(Note 7) System variables 50000 to 51199 are common among part systems.

Variable expressions

Variable:	: # Numerical value (Numerical value: 1, 2, 3, ...)	#100
	: # [Expression]	#100
Expression	: Numerical value	
	: Variable	
	: Expression Operator Expression	#100 + #101
	: - (minus) Expression	-#120
	: [Expression]	[#110]
	: Function [Expression]	SIN [#110]

Variable definition
Variable = expression

(Note 4) Variables cannot be used with addresses "O" and "N".

12.1.2.4.1 100 Sets

M system : ○ **L system : ○**

12.1.2.4.2 200 Sets

M system : Δ **L system : Δ**

12.1.2.4.3 300 Sets

M system : Δ **L system : Δ**

12.1.2.4.4 600 Sets

M system : Δ **L system : Δ**

12.1.2.4.6 (50+50 × Number of Part Systems) Sets

M system : ○ **L system : ○**

12.1.2.4.7 (100+100 × Number of Part Systems) Sets

M system : Δ **L system : Δ**

12.1.2.4.8 (200+100 × Number of Part Systems) Sets

M system : Δ **L system : Δ**

12.1.2.4.9 (500+100 × Number of Part Systems) Sets

M system : Δ **L system : Δ**

12.1.2.101 N Code Macro

M system : Δ **L system : Δ**

This function calls the macro program using a pre-registered N code.
The N No. and the macro program are registered using parameter setting, and up to eight can be registered.
Argument (P, N, L, G) which cannot be used by a usual calling macro can be used.
In addition, the argument G can be used up to four.

(1) Macro call by N code

Format

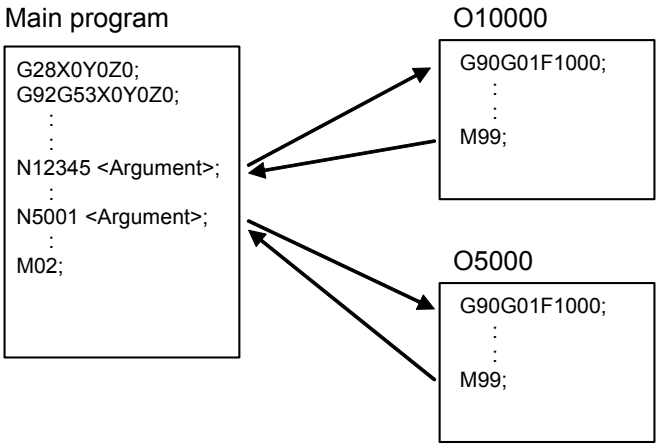
N***** **<Argument> ;**
N***** : N code for performing macro call

(a) The macro is called by N code. (The calling is same as G65.)
(b) The called N No. is registered by the parameter setting. Wild-card (,) can be used for N No. registration.

Example for setting

	<Code>	<Program No.>		
N[01]	12345	10000	…	N No. : 12345, program No. : 10000
N[02]	5,,,	5000	…	To N No. 5000 to 5999, for program No. : 5000

(c) N code call diverges to the macro as the arguments entire address data in same block, and returns to the head of the next block.
(d) The macro subprogram can be called in up to four levels using N code macro call.



(Note 1) When prohibiting the display and edit of the macro program, set the macro program No. to O9000 to O9999, and change properly the parameter setting of edit lock C and the program display lock.

12.1.2.102 Macro Interface Extension (1200 sets)

M system : Δ **L system : Δ**

These variables enable direct reading/writing of the data between CNC machining program and PLC program.

12.1.3 Fixed Cycle

List of fixed cycles

Type of fixed cycle	M system	L system		Remarks
	G code system 1	G code system 2	G code system 3	
Fixed cycle for drilling	G70	G80	G80	Refer to 12.1.3.1. Refer to 4.5.3.
	:	:	:	
	G89	G89	G89	
		G79	G83.2	
	G98	G98	G98	
	G99	G99	G99	
Special fixed cycles	G34			Refer to 12.1.3.2.
	G35	-	-	
	G36			
Fixed cycles for turning machining		G90	G77	Refer to 12.1.3.3.
	-	G92	G78	
		G94	G79	
Multiple repetitive fixed cycles for turning machining		G70	G70	Refer to 12.1.3.4. Refer to 12.1.3.5.
		:	:	
	-	G76	G76	
		G76.1	G76.1	
		G76.2	G76.2	

12.1.3.1 Fixed Cycle for Drilling

M system : O **L system : O**

(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.

G code	Function
G70	
G71	
G72	
G73	Step cycle
G74	Reverse tapping cycle
G75	
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.

G code	Function
G98	Initial point level return
G99	R point level return

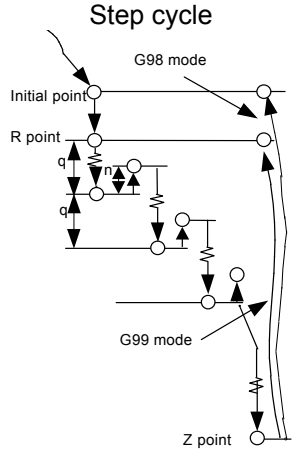
The basic program format for the fixed cycle commands is shown below.

G81 Xx1 Yy1 Zz1 Rr1 Qq1 Pp1 Ll1 Ff1 ;

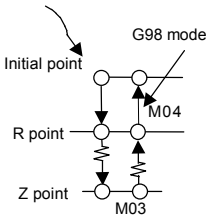
G81	:	Hole drilling mode
Xx1, Yy1	:	Hole position data; X-axis, Y-axis hole drilling position command (rapid traverse) (incremental/absolute)
Zz1	:	Hole machining data; Hole bottom position designation (incremental/absolute)
Rr1	:	Hole machining data; Hole R point designation (incremental/absolute)
Qq1	:	Hole machining data; Depth of cut per pass in G73, G83 cycle (incremental) Shift amount in G76, G87 cycle Depth of cut per pass in pecking tapping, deep hole tapping of G74, G84 cycle
Pp1	:	Hole machining data; Dwell time at hole bottom
Ll1	:	Hole machining data; Number of fixed cycle repetitions
Ff1	:	Cutting feed rate

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

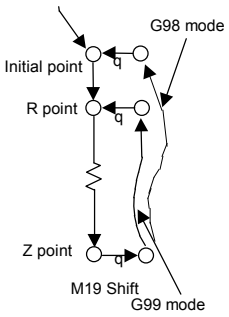
G73
Step cycle



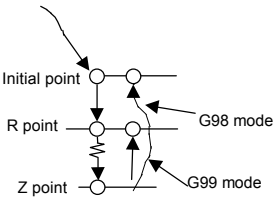
G74
Reverse tapping cycle



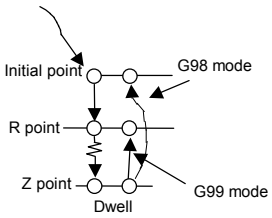
G76
Fine boring cycle



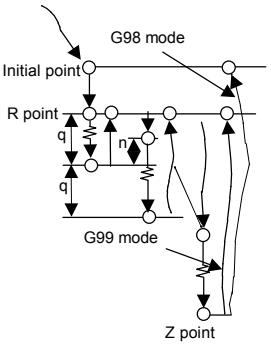
G81
Drilling, spot drilling cycle



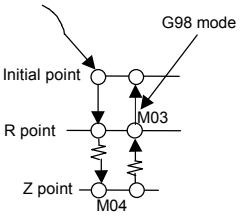
G82
Drilling, counterboring cycle



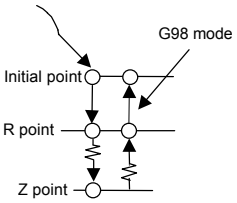
G83
Deep hole drilling cycle



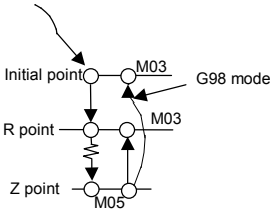
G84
Tapping cycle



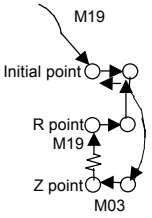
G85
Boring cycle



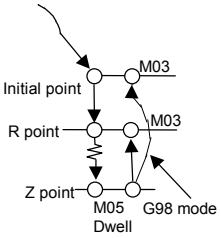
G86
Boring cycle



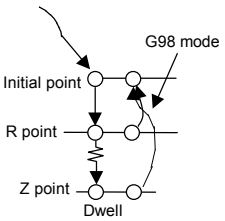
G87
Back boring cycle



G88
Boring cycle



G89
Boring cycle



(2) L system; G83 to G89, G80

In the fixed cycle for drilling, a machining program such as drilling, tapping, or boring and positioning can be executed for a given machining sequence in 1-block commands.

G code	Drilling axis	Drilling work start	Motion at hole bottom	Return 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

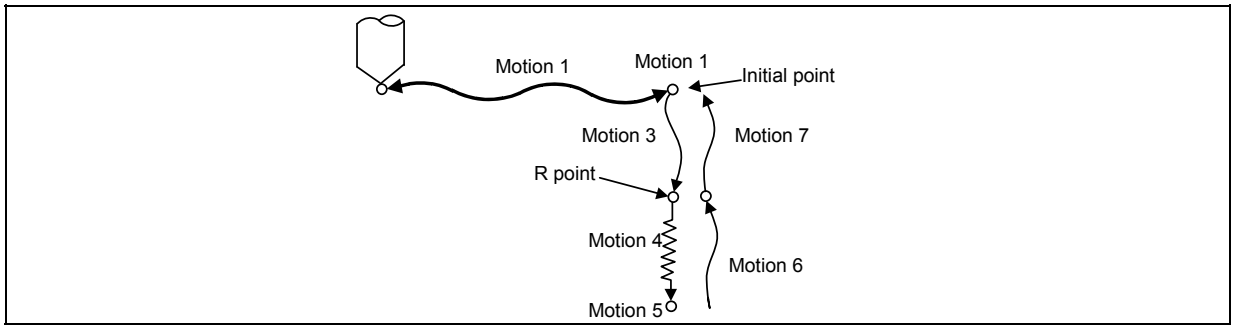
The fixed cycle mode is canceled when a G command of the G80 or G01 group is specified. Data is also cleared simultaneously.

Command format

G83/G84/G85 Xx1 Cc1 Zz1 Rr1 Qq11 Pp1 Ff1 Kk1 (Mm1) Ss1 ,Ss1 Dd1 ,Rr1 ;
G87/G88/G89 Xx1 Cc1 Zz1 Rr1 Qq11 Pp1 Ff1 Kk1 (Mm1) Ss1 ,Ss1 Dd1 ,Rr1 ;

G83/G84/G85	:	Fixed cycle mode of drilling (G83, G87), tapping (G84, G88), or boring (G85, G89)
G87/G88/G89		The drilling command is modal. Once it is given, it is effective until another drill command is given or drilling fixed cycle cancel command 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, Qq11, 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. Hole machining data; R point position (incremental value from initial point) designation (sign ignored).
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 tap 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.
(Operations 6 and 5 may be conducted as a single operation depending on the fixed cycle mode.)

(Note 1) 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:

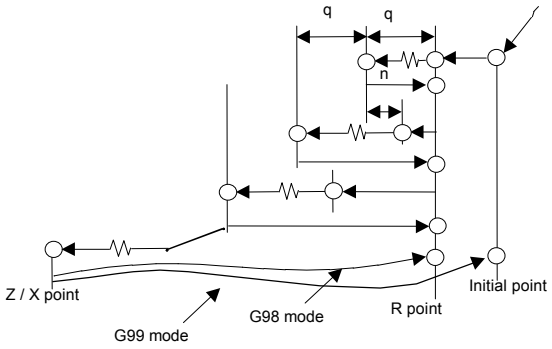
G98: Initial level return
G99: R point level return

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.

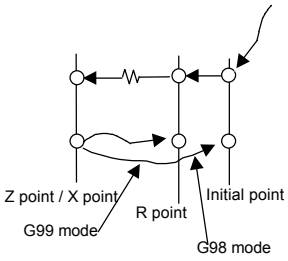
Deep-hole drilling cycle (G83, G87)

G83/G87
Deep-hole drilling cycle (G83: Z-axis direction, G87: X-axis direction)

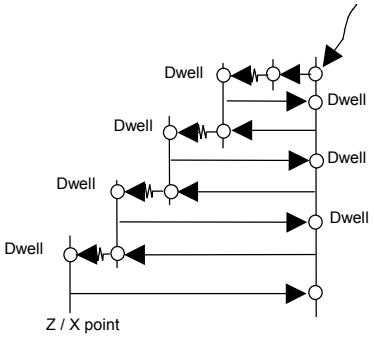
When Q command is given



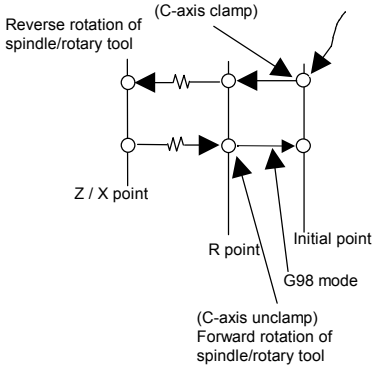
When Q command is not given



G83.2
Deep-hole drilling cycle

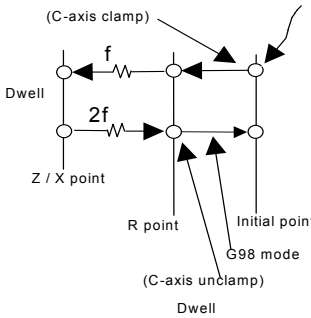


G84/88
Tapping cycle



Output or no output can be set using a parameter for the C-axis clamp/unclamp M code

G85/89
Boring cycle



Output or no output can be set using a paramete
for the C-axis clamp/unclamp M code

There are two levels of hole machining axis return which apply upon completion of the fixed cycle machining operation. (see the figure above)

G code	Function
G98	Initial point level return
G99	R point level return

12.1.3.3 Special Fixed Cycle

M system : Δ **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:

G code	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.

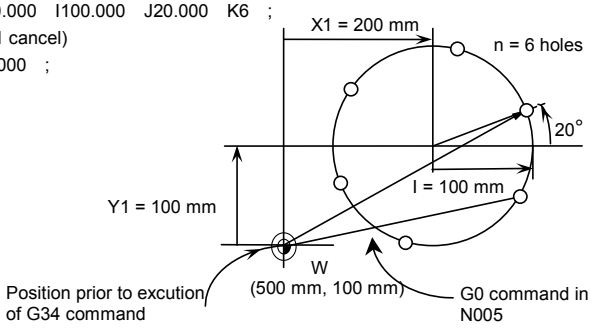
G34 Xx Yy Ir Jθ Kn ;

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 input 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 input 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  ;
```



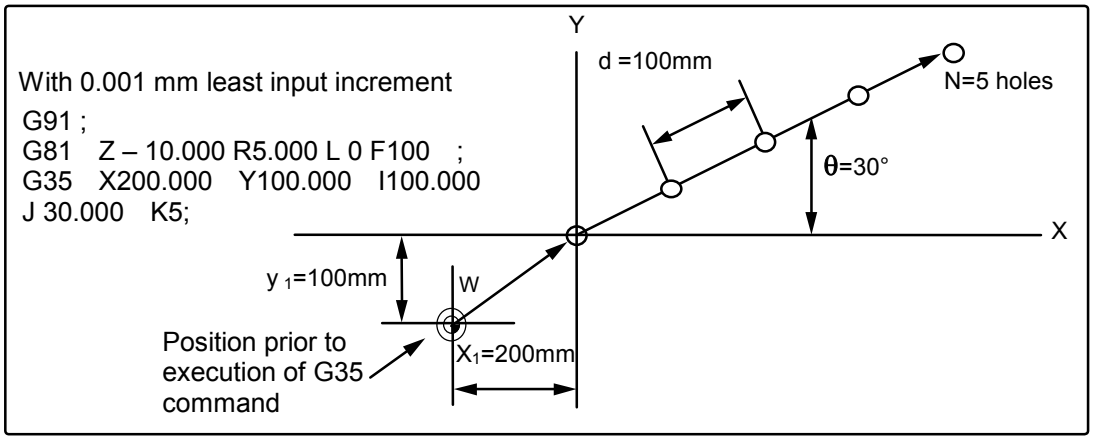
As shown in the figure, the tool is positioned above the final hole upon completion of the G34 command. This means that when it is to be moved to the next position, it will be necessary to calculate the coordinates in order to issue the command or commands with incremental values, and so it is convenient to use the absolute value mode.

(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 Jθ Kn ;		
Xx, Yy	:	The starting point coordinates; they are affected by the G90/G91 commands.
Id	:	Interval "d"; it is based on the least input increment and when "d" is negative, drilling proceeds in the point symmetrical direction centered on the starting point.
Jθ	:	Angle θ; the counterclockwise direction is taken to be positive.
Kn	:	Number "n" of holes to be drilled including the starting point; any number of holes from 1 through 9999 can be assigned.

(Example)



(3) Arc (G36)

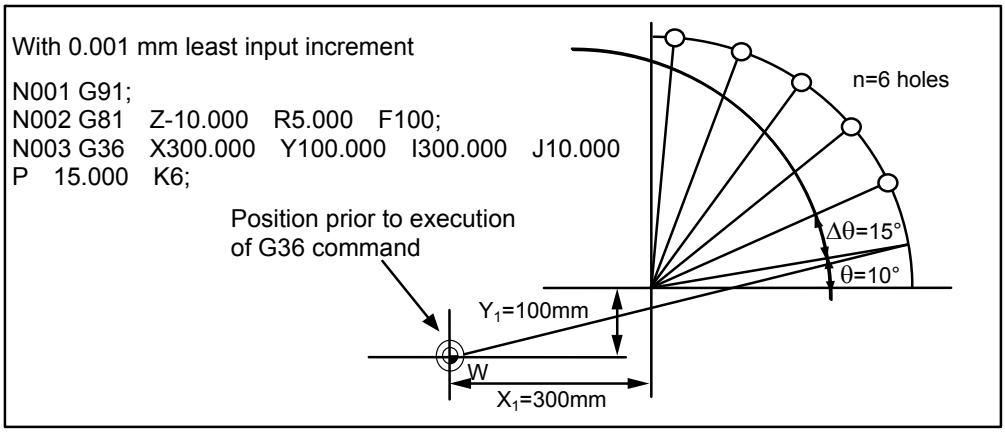
The tool starts at the point forming angle θ with the X axis on the circumference of a circle with radius "r" whose center is the coordinates designated by X and Y, and it drills "n" number of holes aligned at angle interval $\Delta\theta$. As with the bolt hole circle function, the drilling operation 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 input increment and is provided with a positive number.
Jθ	:	Angle θ at the point to be drilled initially; the counterclockwise direction is taken to be positive.
P$\Delta\theta$	:	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.

(Example)

With 0.001 mm least input increment

```
N001 G91;
N002 G81   Z-10.000   R5.000   F100;
N003 G36   X300.000   Y100.000   I300.000   J10.000
P   15.000   K6;
```

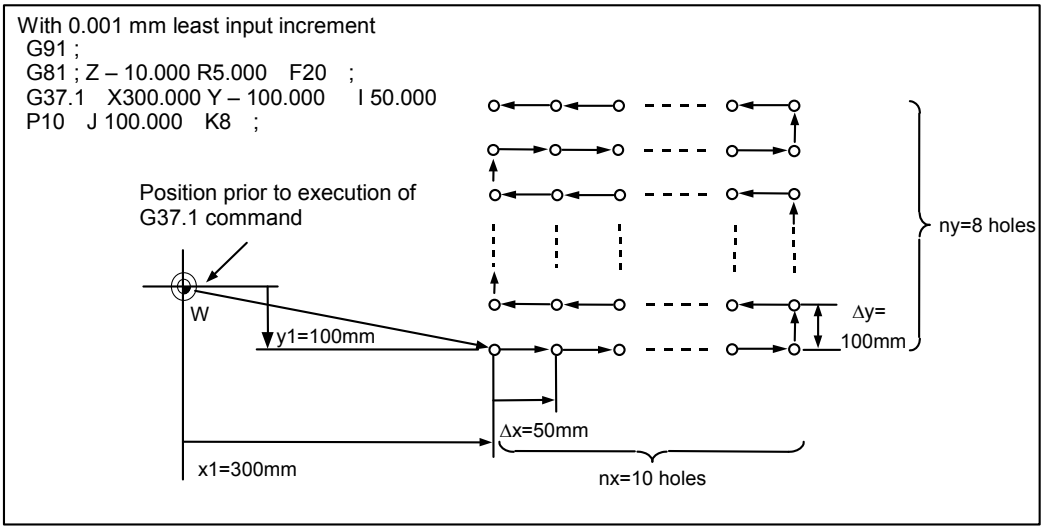


(4) Grid (G37.1)

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 ∆x 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 Xx1 Yy1 I∆x Pnx J∆y Kny ;		
Xx, Yy	:	The starting point coordinates; they are affected by the G90/G91 commands.
I∆x	:	X-axis interval ∆x; it is based on the least input increment; when ∆x 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∆y	:	Y-axis interval ∆y; it is based on the least input increment; when ∆y 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.

(Example)



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∆∆ 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:

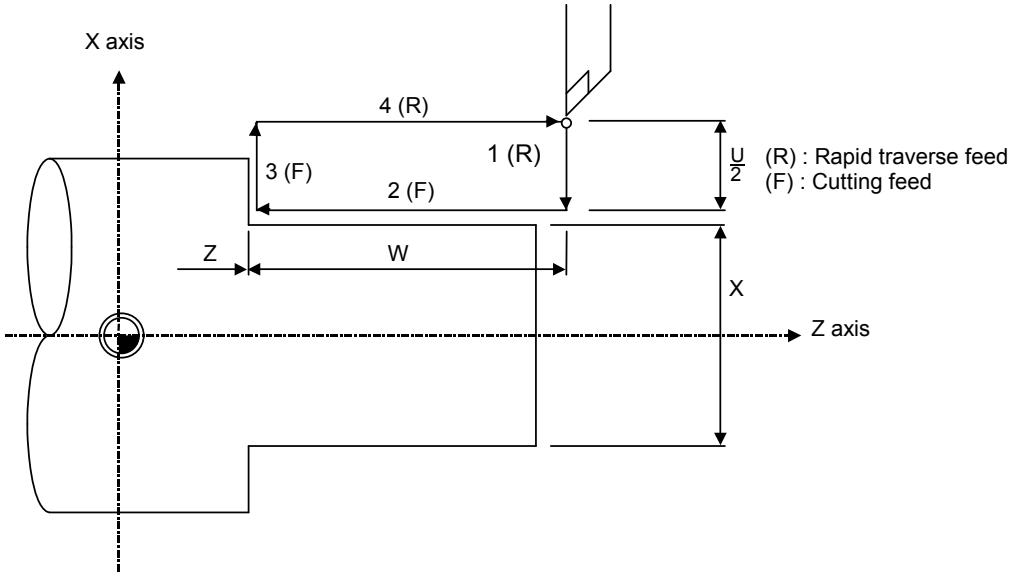
G00, G01, G02, G03
G09
G10, G11
G27, G28, G29, G30
G31
G33, G34
G37
G92
G52, G53
G65

(1) Longitudinal cutting cycle (G77)

(a) Longitudinal cutting

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

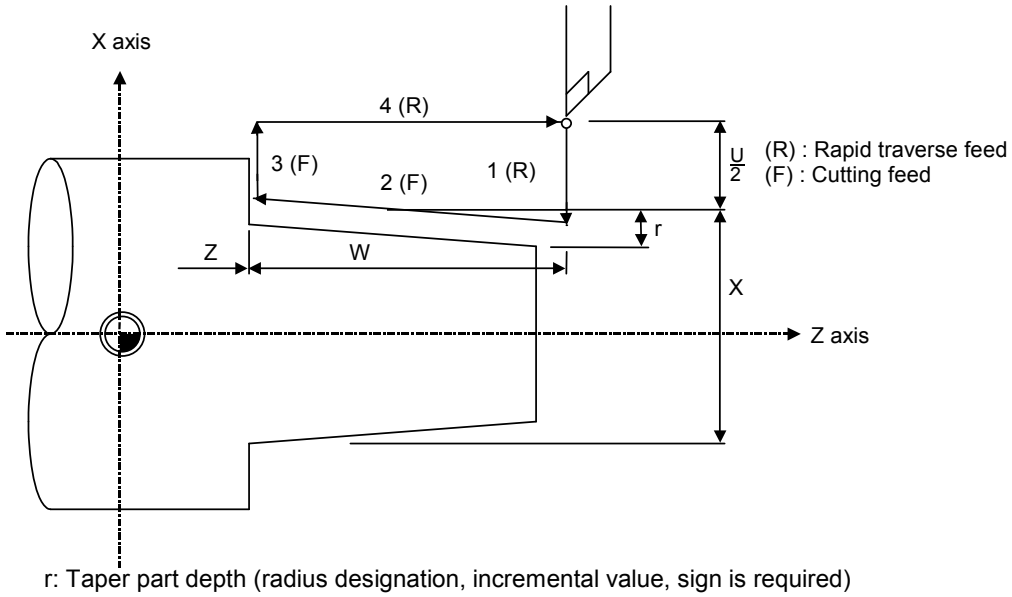
G77 X/U_ Z/W_ F_ ;



(b) Taper cutting

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

G77 X/U_ Z/W_ R_ F_ ;



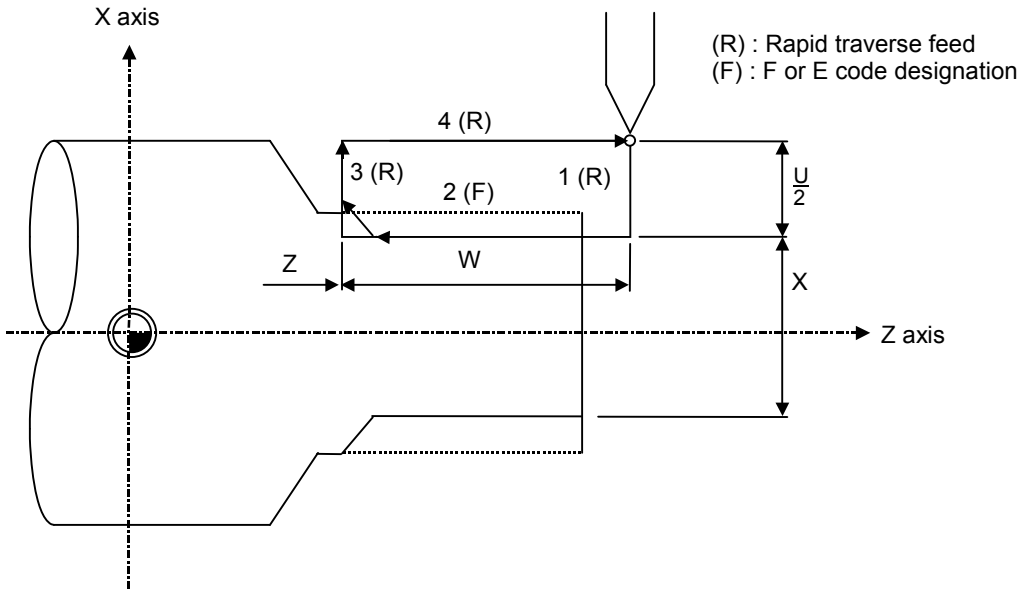
r: Taper part depth (radius designation, incremental value, sign is required)

(2) Thread cutting cycle (G78)

(a) Straight thread cutting

Straight thread cutting can be performed by the following block:

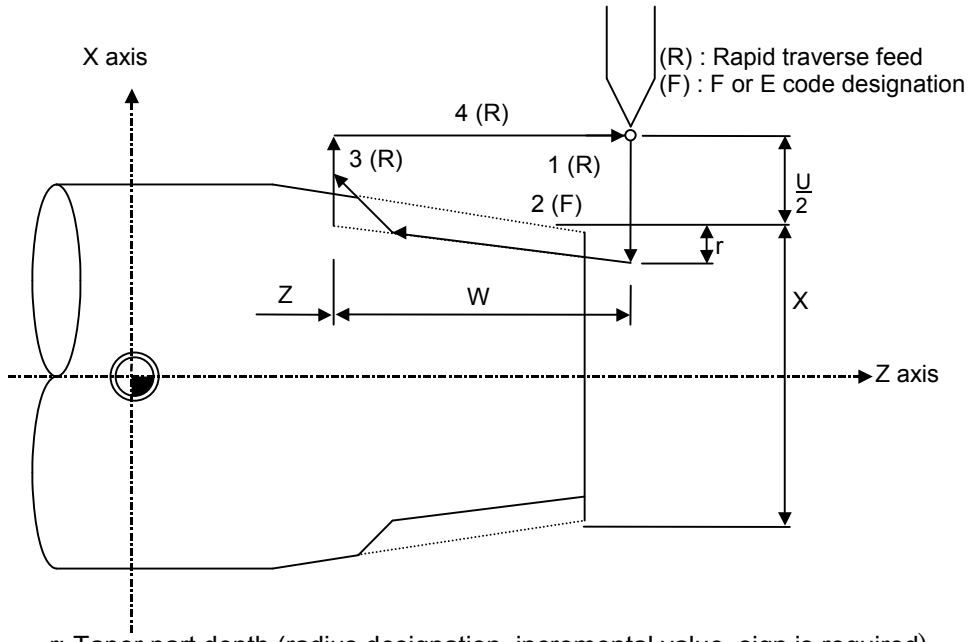
G78 X/U_ Z/W_ F/E_ ;



(b) Taper thread cutting

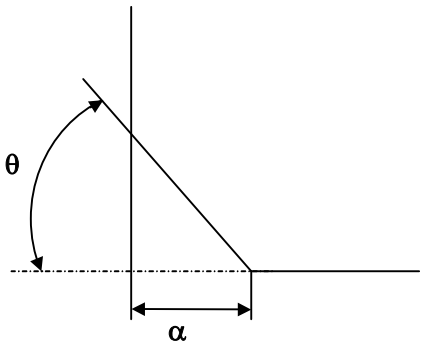
Taper thread cutting can be performed by the following block:

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



r: Taper part depth (radius designation, incremental value, 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.

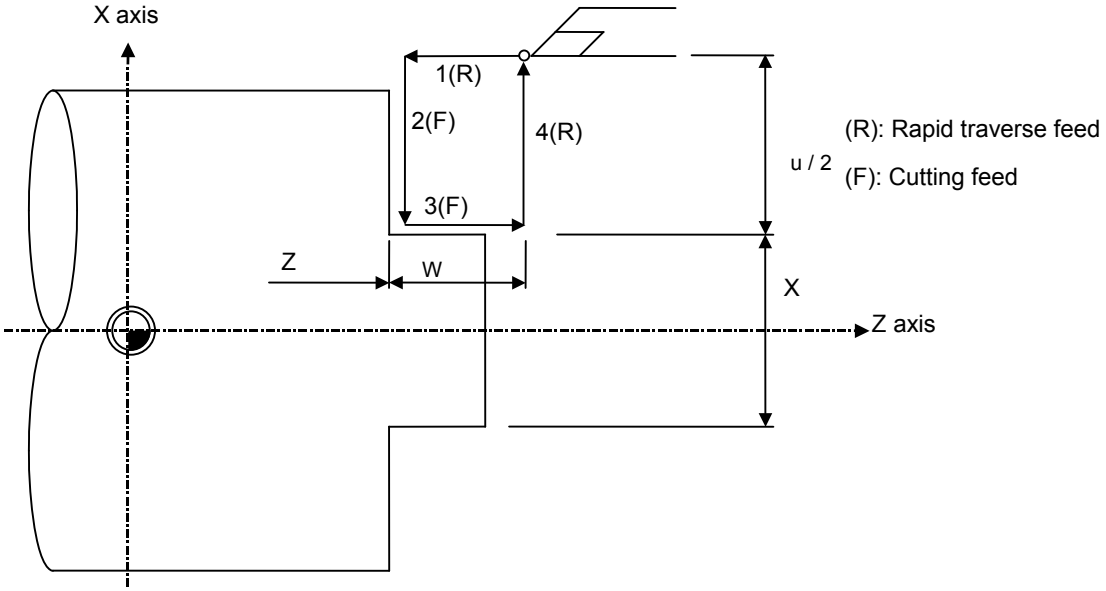
θ : Thread cutting-up angle
The thread cutting-up angle can be set in a given parameter in 1° steps in the range of 0 to 89°.

(3) Face cutting cycle (G79)

(a) Straight cutting

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

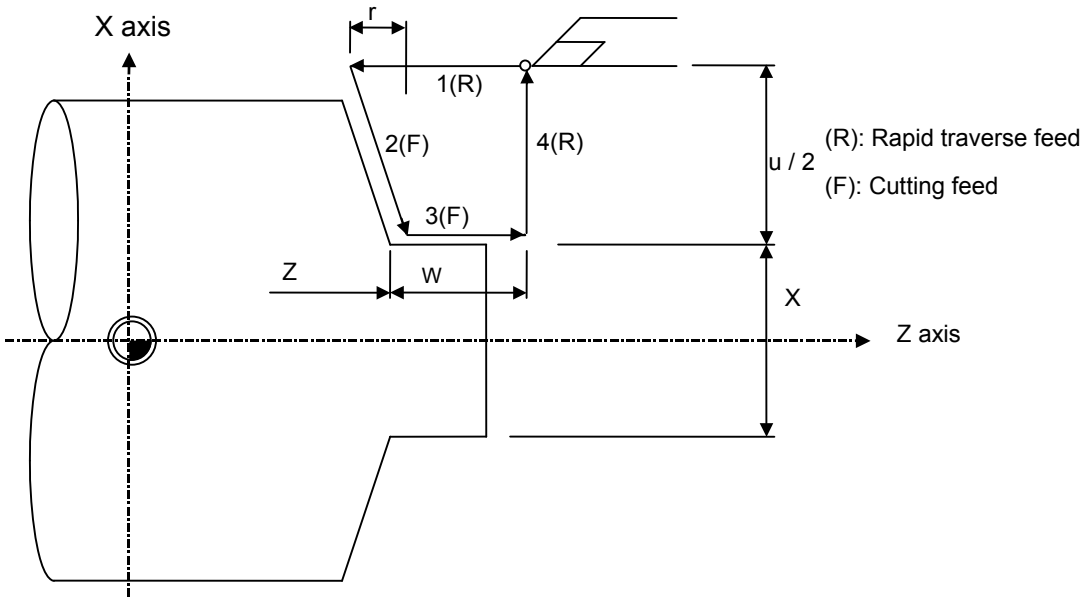
G79 X/U_ Z/W_ F_ ;



(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: Taper part depth (radius designation, incremental value, 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)

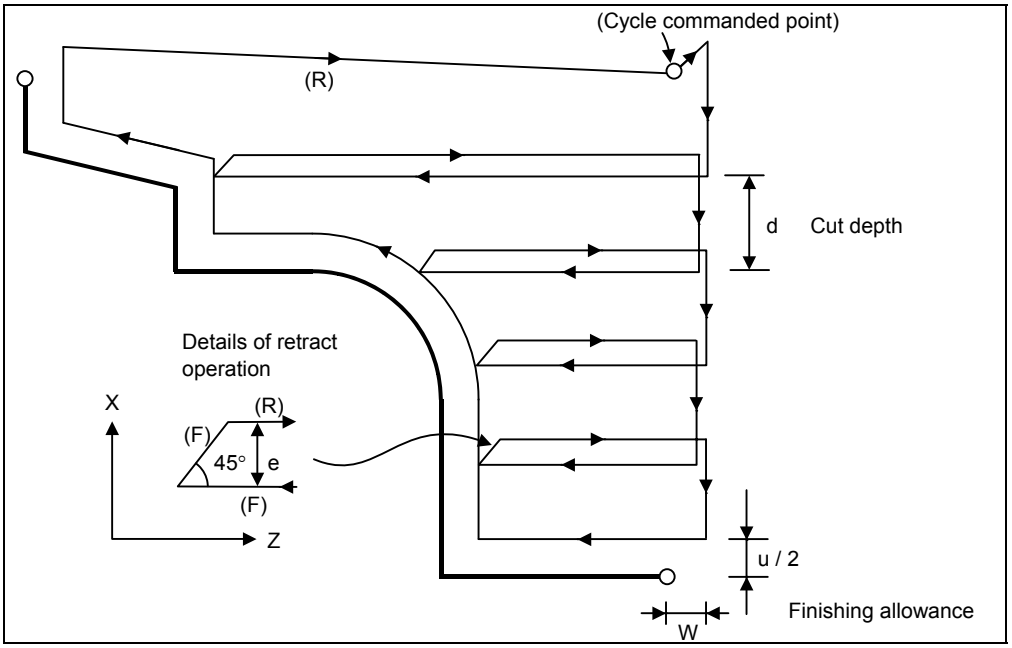
The finish shape program is called, and straight rough cutting is performed while intermediate path is being calculated automatically.

The machining program is commanded as follows.

G71 Ud Re ;
G71 Aa Pp Qq Uu Ww Ff Ss Tt ;

Ud : Cut depth d. (When P,Q command is not given). (Modal)
Re : Retract amount e. (Modal)
Aa : Finish shape program No. (If it is omitted, the program being executed is assumed to be designated.)
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.
Uu : Finishing allowance in the X axis direction. (When P, Q command is given). (Diameter or radius designation)
Ww : Finishing allowance in the Z axis direction.
Ff : Cutting feed rate.
Ss : Spindle speed.
Tt : Tool command.

} F, S, and T command in the finish shape program are ignored, and the value in the rough cutting command or the preceding value becomes effective.


(Cycle commanded point)
(R)
d
Cut depth
Details of retract operation
X
(R)
(F)
45°
e
(F)
Z
u / 2
Finishing allowance
W

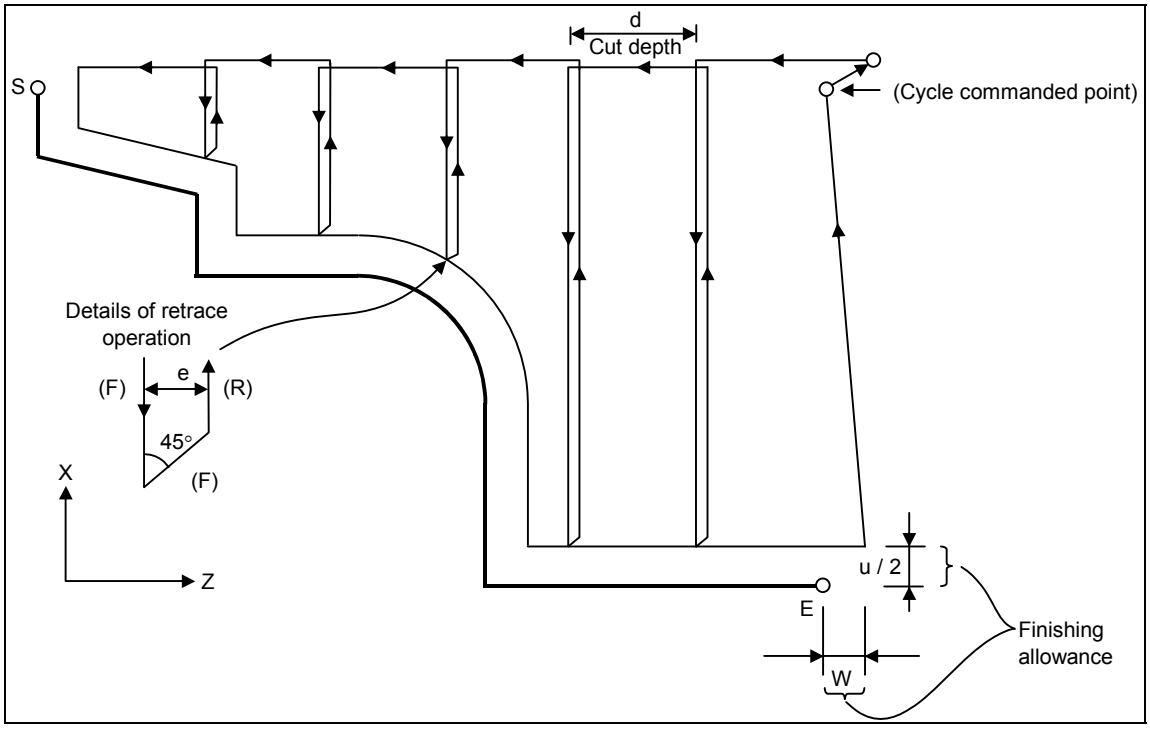
(2) Face rough cutting cycle (G72)

The finish shape program is called, and rough turning 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 ;

Wd	: Cut depth d. (When P,Q command is not given). (Modal)	
Re	: Retract amount e. (Modal)	
Aa	: Finish shape program No. (If it is omitted, the program being executed is assumed to be designated.)	
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.	
Uu	: Finishing allowance in the X axis direction.	
Ww	: Finishing allowance in the Z axis direction. (When P, Q command is given.)	
Ff	: Cutting feed rate.	F, S, and T command in the finish shape program are ignored, and the value in the rough cutting command or the preceding value becomes effective.
Ss	: Spindle speed.	
Tt	: Tool command.	



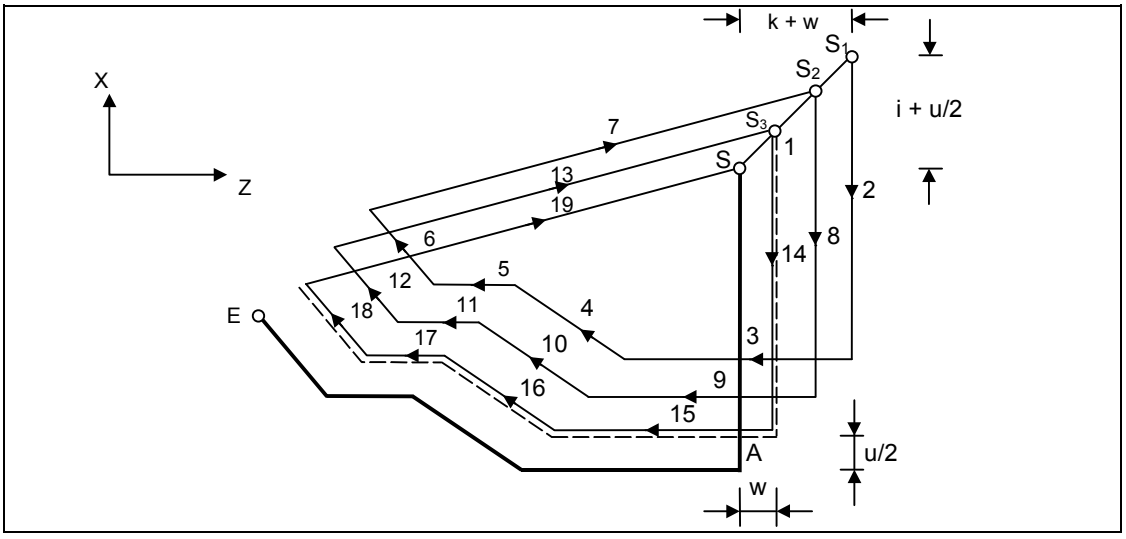
(3) Molding material in rough cutting cycle (G73)

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. • Modal data • Sign is ignored. • Cutting allowance is given with a radius designation.
Wk	: Cutting allowance in the Z axis direction k	
Rd	: Split count d	
Aa	: Finish shape program No.	(If it is omitted, the present program is assumed to be designated.)
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 Qq command, up to M99.
Uu	: Finishing allowance in the X axis direction u	• Finishing allowance when P, Q command is given. • Sign is ignored. • Diameter or radius is designated according to the parameter. • The shift direction is determined by the shape.
Ww	: Finishing allowance in the Z axis direction w	
Ff	: Cutting 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 turning can be performed by using the G70 command.

The machining program is commanded as follows.

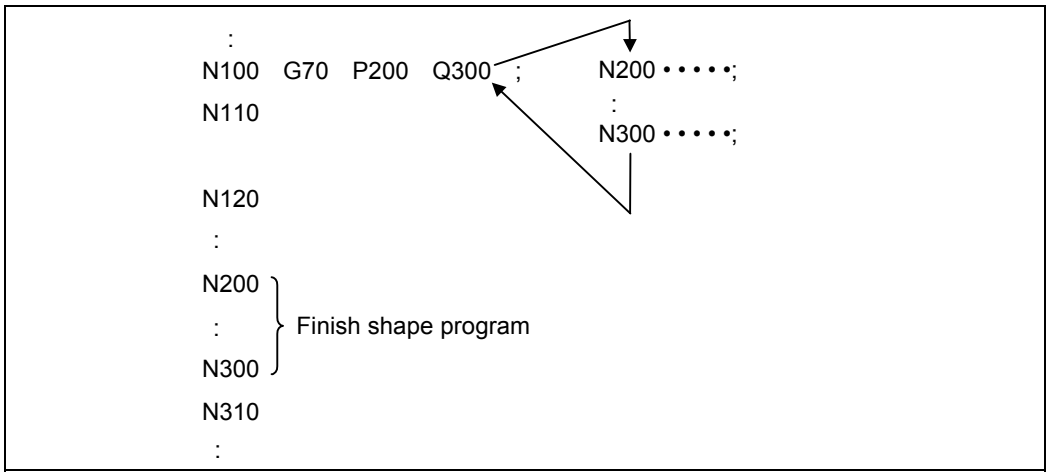
G70 A_ P_ Q_ ;	
A	: Finish shape program number. (If it is omitted, the program being executed is assumed to be designated.)
P	: Finish shape start sequence number. (If it is omitted, the program top is assumed to be designated.)
Q	: Finish shape end sequence number. (If it is omitted, the program end 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.

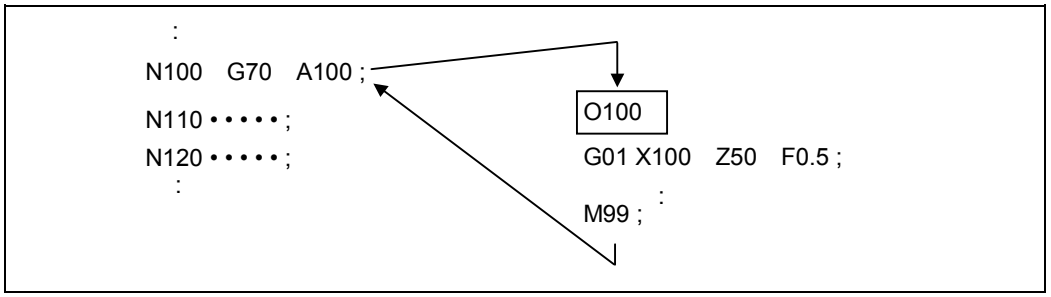
(b) The memory address of the finish shape program executed by G71 to G72 is not stored. Whenever G70 is executed, a program search is made.

(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.

(Example 1) Sequence No. designation



(Example 2) Program No. designation



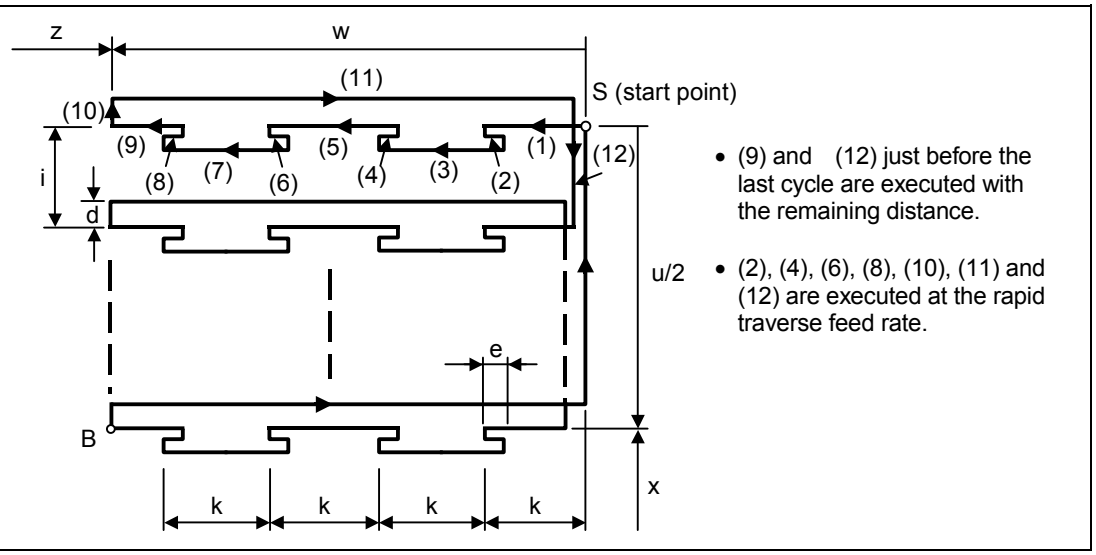
In either example 1 or 2, after the N100 cycle is executed, the N110 block is executed.

(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 X/(U) Z/(W) Pi Qk Rd Ff ;

Re	:	Retract amount e (when X/U, Z/W command is not given) (Modal)
X/U	:	B point coordinate (absolute/incremental)
Z/W	:	B point coordinate (absolute/incremental)
Pi	:	Tool shift amount (radius designation, incremental, sign not required)
Qk	:	Cut depth k (radius designation, incremental, 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 at the second cut bottom and later.)
Ff	:	Feed rate

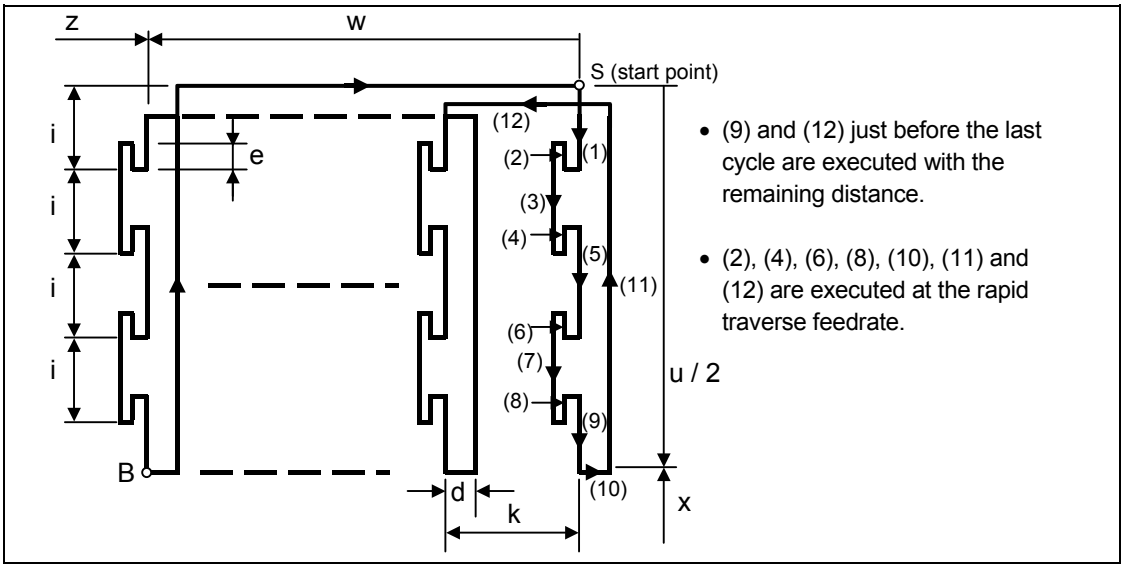


- (9) and (12) just before the last cycle are executed with the remaining distance.
- (2), (4), (6), (8), (10), (11) and (12) are executed at the rapid traverse feed rate.

(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	**X/(U) Z/(W) Pi Qk Rd Ff ;**	
Re	:	Retract amount e (when X/U, Z/W command is not given) (Modal)
X/U	:	B point coordinate (absolute/incremental)
Z/W	:	B point coordinate (absolute/incremental)
P	:	Tool shift amount (radius designation, incremental, sign not required)
Qk	:	Cut depth k (radius designation, incremental, sign not required)
Rd	:	Relief amount at cut bottom d (If sign is not provided, relief is made at the first cut bottom. If ⊖ sign is provided, relief is made not at the first cut bottom but at the second cut bottom and later.)
Ff	:	Feed rate



(7) Multiple repetitive thread cutting cycle (G76)

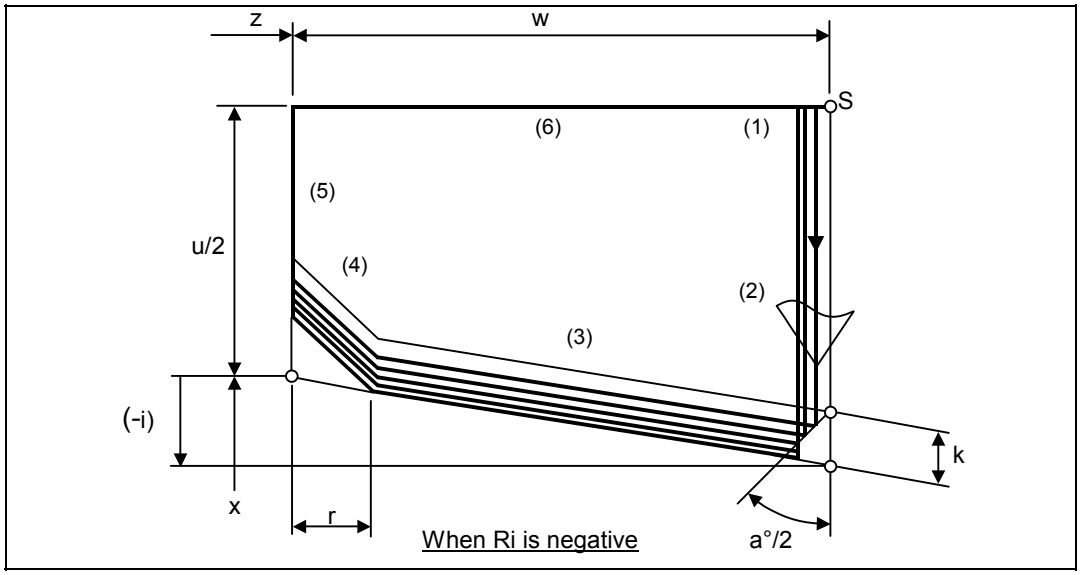
When the thread cutting start and end points are commanded, cut at any desired angle can be made by automatically cutting so that the cut section area (cutting torque) per time becomes constant in the G76 fixed cycle.

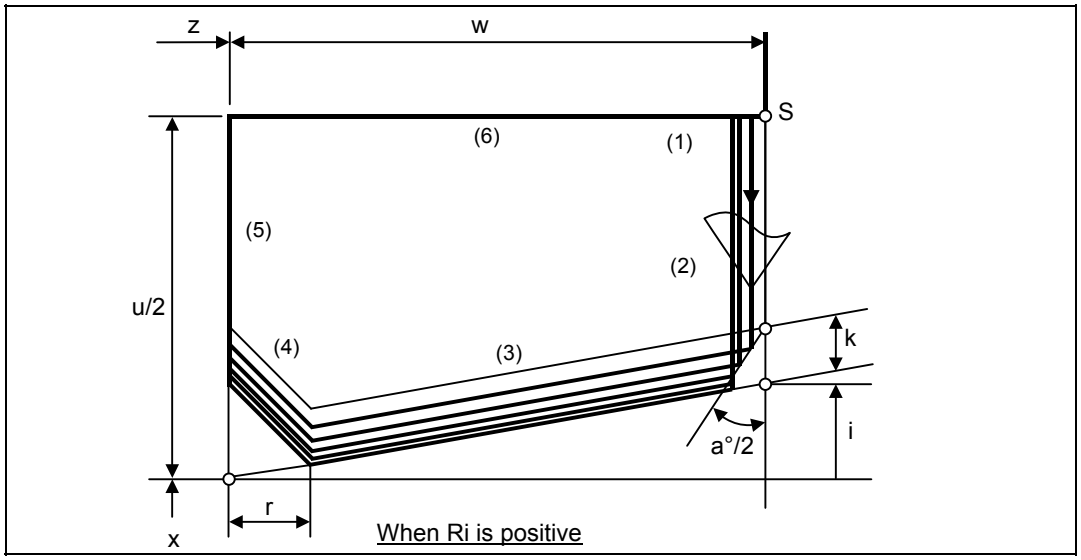
Various longitudinal threads can be cut by considering the thread cutting end point coordinate and taper height constituent command value.

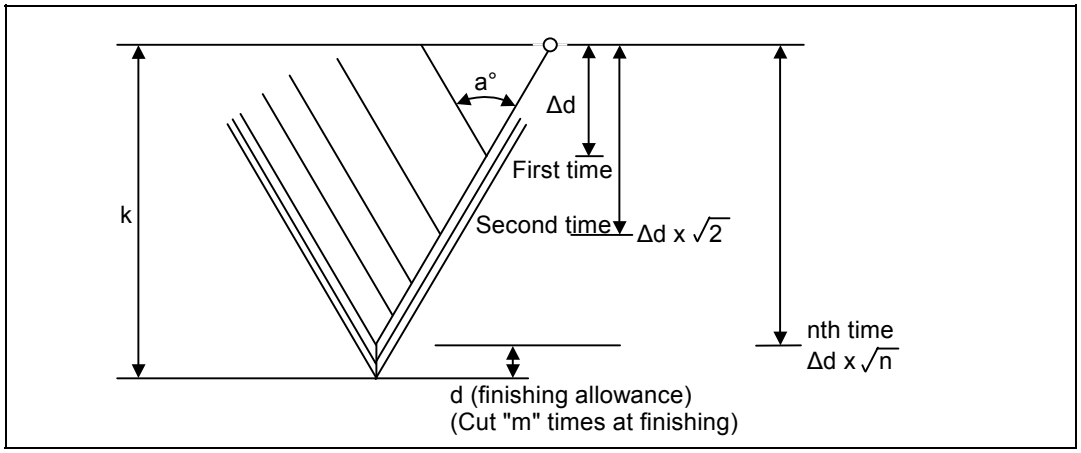
Command Format

G76	**Pmra**	**Rd ;**
G76	**X/U Z/W Ri Pk QΔd Fl**	**;**
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.
d	:	Finishing allowance (modal)
X/U	:	X axis end point coordinate of thread part. Designate the X coordinate of the end point in the thread part in an absolute or incremental value.
Z/W	:	Z axis end point coordinate of thread part. Designate the Z coordinate of the end point in the thread part in an absolute or incremental value.
i	:	Taper height constituent in thread part (radius value). When i = 0 is set, straight screw is made.
k	:	Thread height. Designate the thread height in a positive radius value.
Δd	:	Cut depth. Designate the first cut depth in a positive radius value.
l	:	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.







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 an image, this function can machine the other side of the image 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.

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

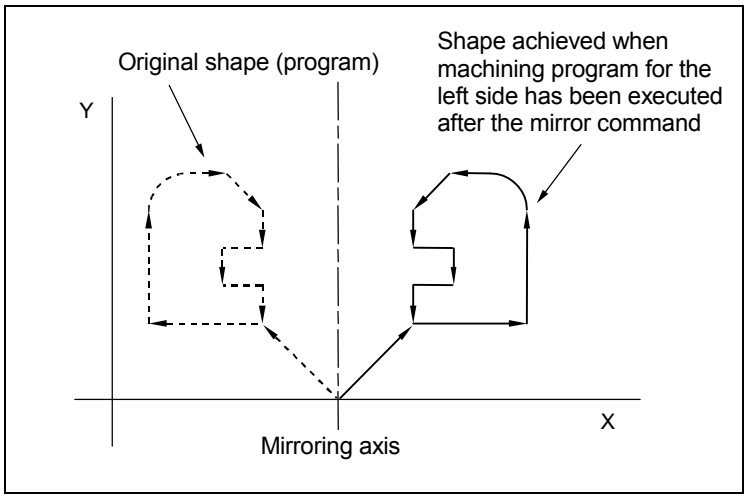
With the local coordinate system, the mirror image is applied with the mirror positioned respectively at x1, y1 and z1.

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 coordinate word indicates the axes for which the mirror image function is to be canceled and the coordinates are ignored.

In the case of G51.1 Xx1



12.1.4.4 Mirror Image for Facing Tool Posts

M system : - **L system : Δ**

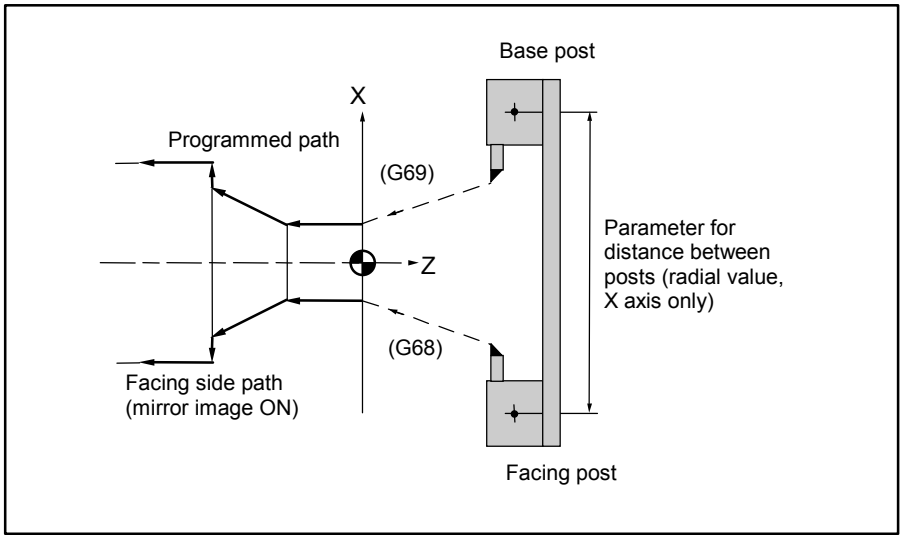
With machines in which the base tool post and facing tool post are integrated, this function enables the programs prepared for cutting at the base side to be executed by the tools on the facing side.
The distance between the two posts is set beforehand with the parameter.

The command format is given below.

G68;	Facing tool post mirror image ON
G69;	Facing tool post mirror image OFF

When the G68 command is issued, the subsequent program coordinate systems are shifted to the facing side and the movement direction of the X axis is made the opposite of that commanded by the program.
When the G69 command is issued, the subsequent program coordinate systems are returned to the base side.

The facing tool post mirror image function can be set to ON or OFF automatically by means of T (tool) commands without assigning the G68 command.
A parameter is used to set ON or OFF for the facing tool post mirror image function corresponding to the T commands.



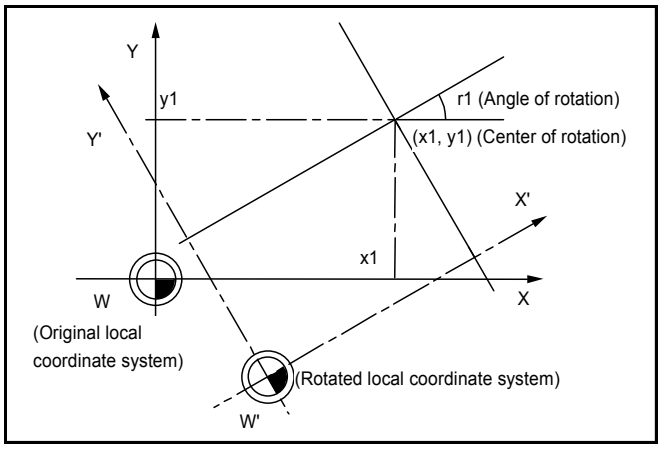
12.1.5 Coordinate System Operation

12.1.5.1 Coordinate Rotation by Program

M system : Δ **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, and 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.

G68 Xx1 Yy1 Rr1 ;	Coordinate rotation ON
G69 ;	Coordinate rotation cancel
G68	: Coordinate rotation command
Xx1, Yy1	: Rotation center coordinates
Rr1:	: Angle of rotation



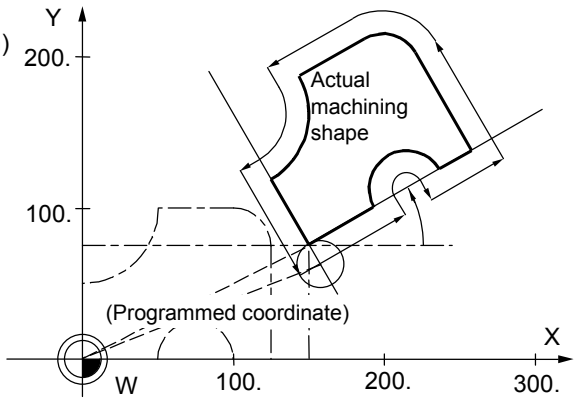
(1) Angle of rotation "r1" can be set in least input increment from –360° to 360°.
(2) The coordinates are rotated counterclockwise by an amount equivalent to the angle which is designated by angle of rotation "r1".
(3) The counter is indicated as the point on the coordinate system prior to rotation.
(4) The rotation center coordinates are assigned with absolute values.

(Example)

```
N01 G28 X Y Z ;
N02 G54 G52 X150. Y75. ;   Local coordinate system assignment
N03 G90 G01 G42 X0 Y0 ;    Tool radius compensation ON
N04 G68 X0 Y0 R30. ;       Coordinate rotation ON
N05 M98 H101 ;             Subprogram execution
N06 G69 ;                  Coordinate rotation cancel
N07 G54 G52 X0 Y0 ;        Local coordinate system cancel
N08 G00 G40 X0 Y0 ;        Tool radius compensation cancel
N09 M02 ;                  Completion
```

Sub program
(Shape programmed with original coordinate system)

```
N101 G90 G01 X50. F200 ;
N102 G02 X100. R25. ;
N103 G01 X125. ;
N104 Y75. ;
N105 G03 X100. Y100. R25. ;
N106 G01 X50. ;
N107 G02 X0 Y50. R50. ;
N108 G01 X0 Y0 ;
N109 M99 ;
```



12.1.6 Dimension Input

12.1.6.1 Corner Chamfering/Corner R

M system : Δ **L system : Δ**

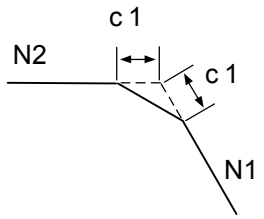
This function executes corner processing by automatically inserting a straight line or arc in the commanded amount between two consecutive movement blocks (G01/G02/G03).
The corner command is executed by assigning the ",C" or ",R" command for the block at whose end point the corner is inserted.

(1) Corner chamfering / 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**

Example:

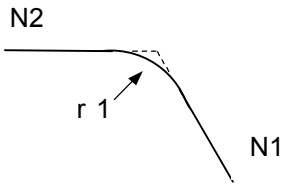
```
N1 G01 Xx1 Zz1,  Cc1  ;
N2 Zz2  ;
```



- **Corner R**

Example:

```
N1 G01 Xx1 Zz1,  Rr1  ;
N2 Zz2  ;
```

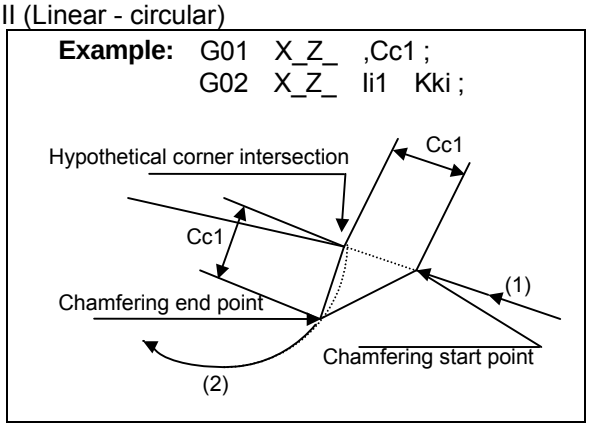


(Note 1) If a corner chamfering or corner R command is issued specifying a length longer than the N1 or N2 block, a program error occurs.

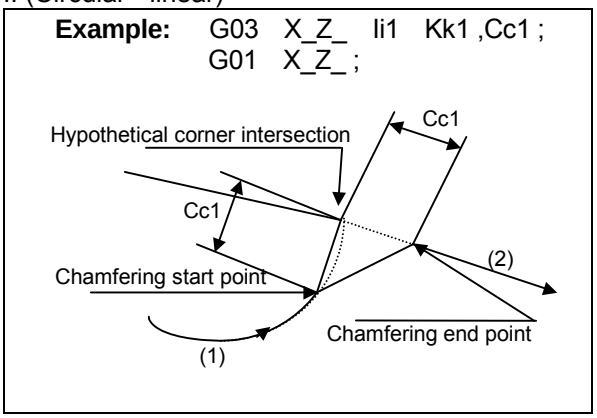
(2) Corner chamfering / corner R II (L system)

When ",C" or ",R" is command in a program between linear-circular, corner chamfering or corner R can be inserted between blocks.

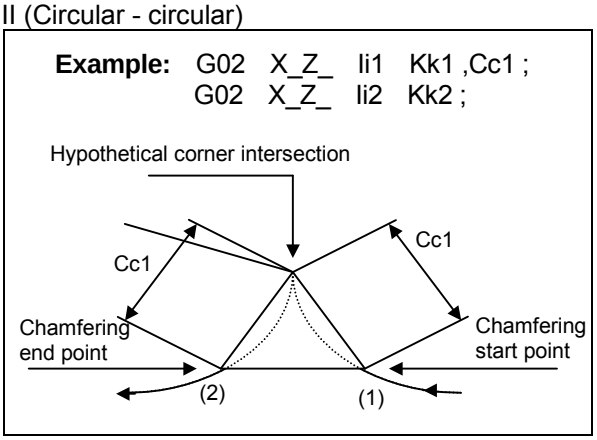
(a) Corner chamfering II (Linear - circular)



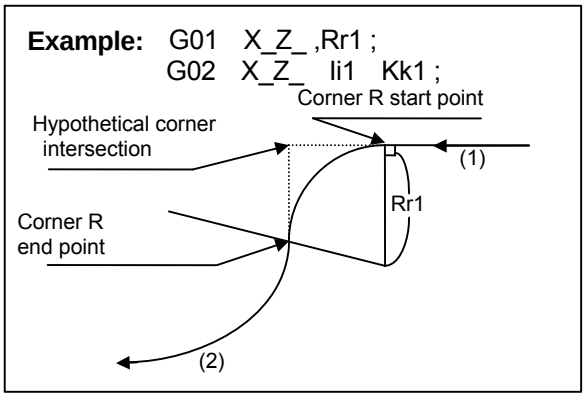
(b) Corner chamfering II (Circular - linear)



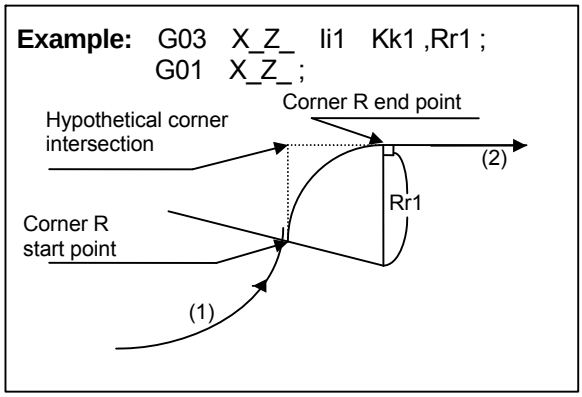
(c) Corner chamfering II (Circular - circular)



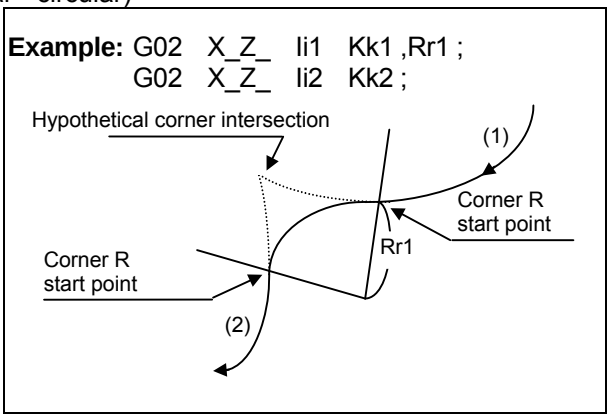
(d) Corner R II (Linear - circular)



(e) Corner R II (Circular - linear)



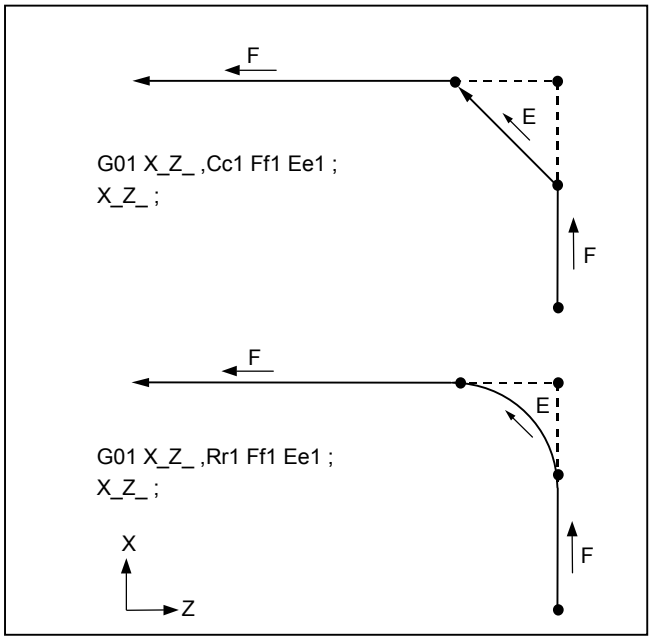
(f) Corner R II (Circular - circular)



(3) Specification of corner chamfering / corner R speed E

An E command 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)



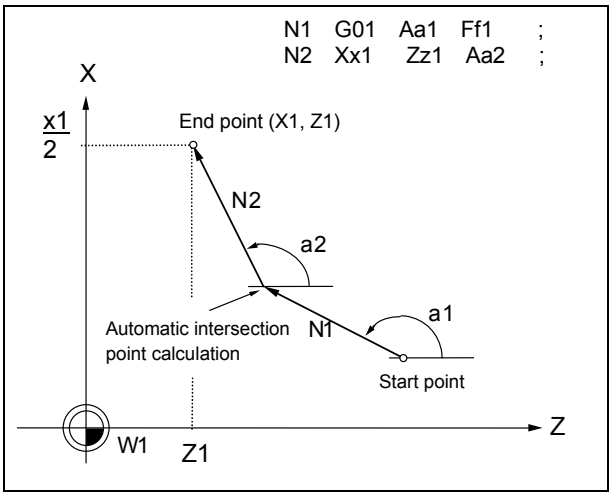
An E command is a modal and remains effective for feeding in next corner chamfering or corner R.
An E command has two separate modals: synchronous and asynchronous feed rate modals. The effective feed rate is determined by synchronous (G95) or asynchronous (G94) mode.
If an E command is specified in 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.
Hold or non-hold can be selected (M system only) using a parameter for the E command modal at the time of resetting. It is cleared when the power is turned OFF (as it is with an F command).

12.1.6.3 Geometric Command

M system : - **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 point.

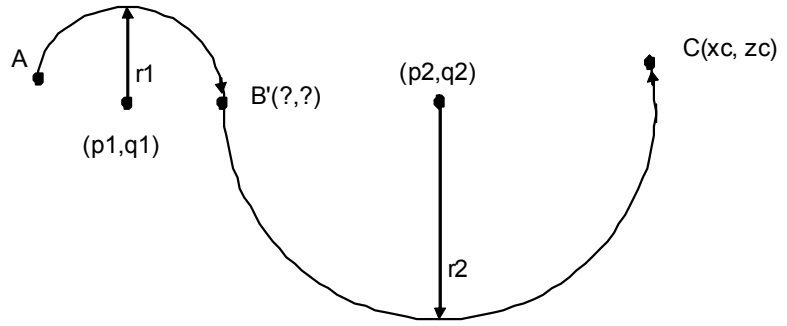
(Note 1) This function cannot be used when using the A axis or 2nd miscellaneous function A.

(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 or radius of the first circular arc and the end point absolute coordinates and center coordinates or radius of the second circular arc.

Example

```
G18   G02   Ii1   Kk1   Ff1   ;
      G03   Xxc   Zzx   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   ;
```



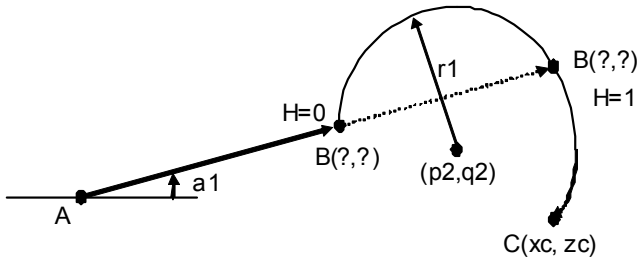
I and K are circular center coordinate incremental values; distances from the start point in the first block or distances from the end point in the second block. P and Q commands (X, Z absolute center coordinates of circular arc) can be given instead of I and K commands.

(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   Ff2   ;
```



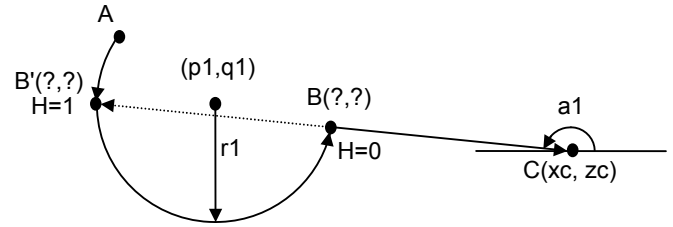
I and K : Incrimental coordinates from circular end point
P and Q : Absolute center coordinates of circular arc
H = 0 : Intersection with shoter line
H = 1 : Intersection with longer line

(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  Ff2  ;
```



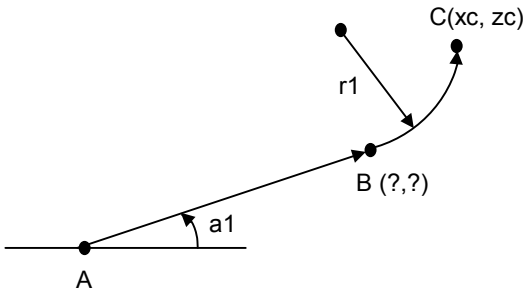
I and K : Incrimental coordinates from circular end point
P and Q : Absolute center coordinates of circular arc
H = 0 : Intersection with shoter line
H = 1 : Intersection with longer line

(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  Ff2  ;
```

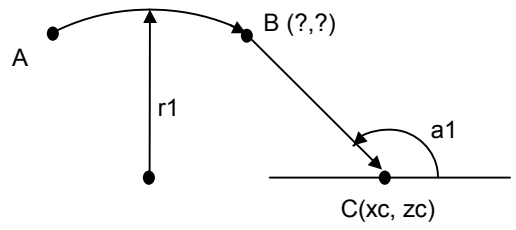


(5) Automatic calculation of arc-linear contact

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  Ff2  ;
```



12.1.7 Axis Control

12.1.7.1 Chopping

12.1.7.1.1 Chopping

M system : Δ **L system : Δ**

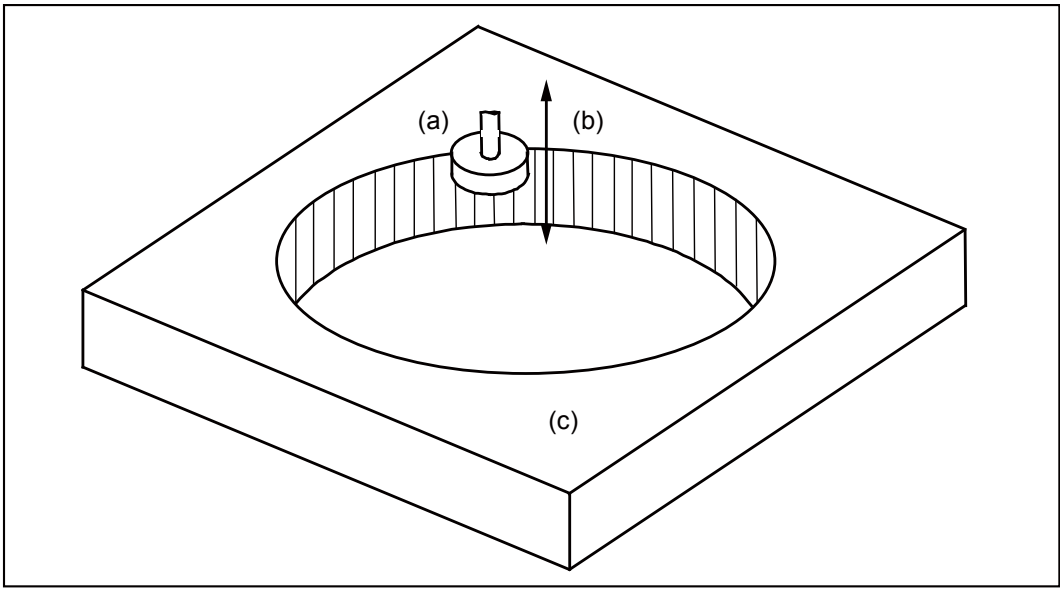
With this function, the chopping axis constantly moves back and forth independently of the program operation during executing the machining program. During the grinding operation, chopping can produce a better surface accuracy than using abrasive grain.

G81.1 Z__ Q__ F__ ; Starting the chopping operation

Z_ : The upper dead point (Select the chopping axis with commanded axis address)
Q_ : Command the distance between the upper dead point and the lower dead point with incremental value
F_ : Command the feedrate during chopping (mm/min)

G80 ; Cancelling the chopping operation

This function continuously raises and lowers the chopping axis independently of program operation when workpiece contours are to be cut.
There are two types of commands for the chopping function: a command by the machining program and a command by a signal from the PLC. Use "#1323 chopsel (chopping command method)" to select which command to use for this function.



(a) Grindstone
(b) Chopping action
(c) Workpiece

12.1.7.3 Circular Cutting

M system : Δ **L system : -**

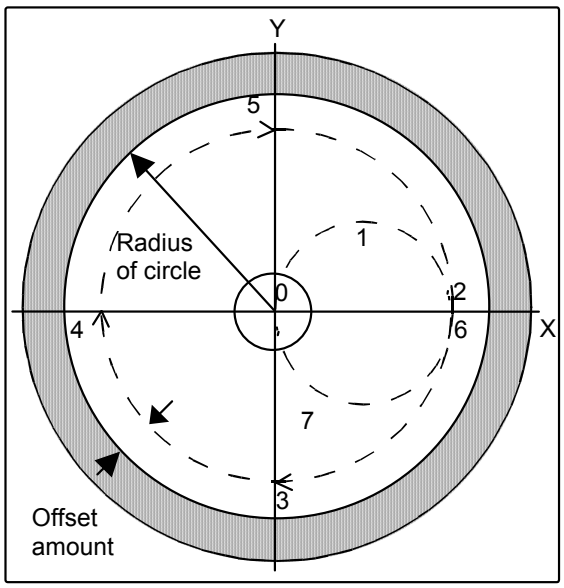
A series of cuts is performed: first, the tool departs from the center of the circle, and by cutting along the inside circumference of the circle, it draws a complete circle, then it returns to the center of the circle. The position at which G12 or G13 has been programmed serves as the center of the circle.

G code	Function
G12	CW (clockwise)
G13	CCW (counterclockwise)

The program format is given below.

G12/13 Ii Dd Ff ;

G12/13 : Circular cutting command
Ii : Radius of complete circle
Dd : Compensation number
Ff : Feed rate



When the G12 command is used
(path of tool center)
$0 \rightarrow 1 \rightarrow 2 \rightarrow 3 \rightarrow 4 \rightarrow 5 \rightarrow 6 \rightarrow 7 \rightarrow 0$
When the G13 command is used
(path of tool center)
$0 \rightarrow 7 \rightarrow 6 \rightarrow 5 \rightarrow 4 \rightarrow 3 \rightarrow 2 \rightarrow 1 \rightarrow 0$

(Notes)

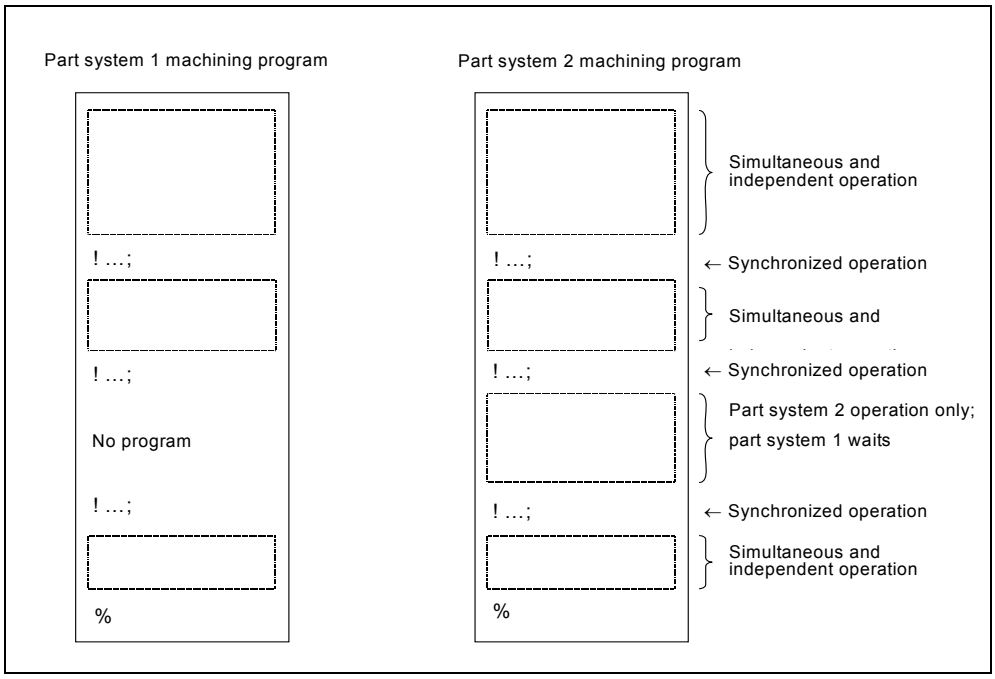
- Circular cutting is undertaken on the plane which has been currently selected (G17, G18 or G19).
- The (+) and (–) signs for the compensation amount denote reduction and expansion respectively.

12.1.8 Multi-part System Control

12.1.8.1 Timing Synchronization Between Part Systems

M system : O **L system : O**

The multi-axis, multi-part system compound control CNC system can simultaneously run multiple machining programs independently. This function is used in cases when, at some particular point during operation, the operations of different part systems are to be synchronized or in cases when the operation of only one part system is required.

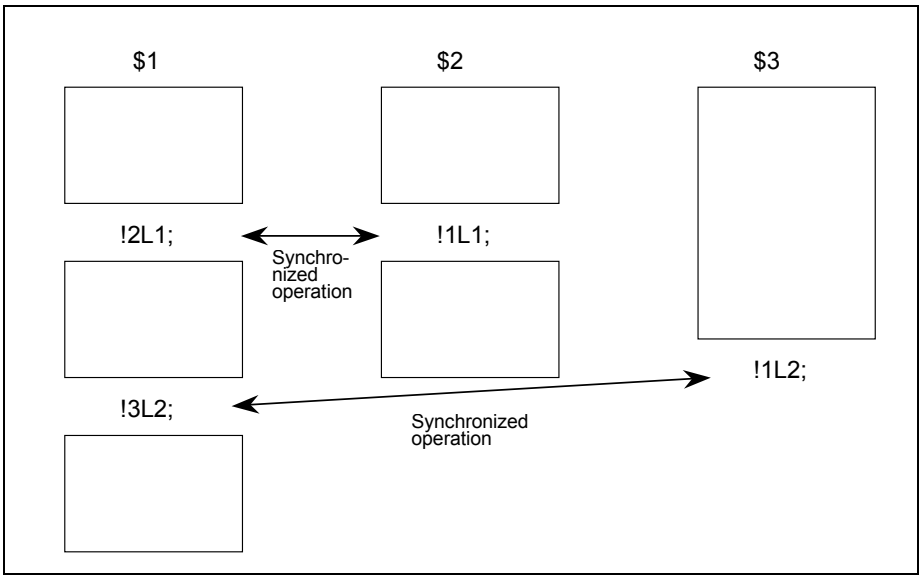


Command format

(1) Command for synchronizing with part system n

!nL1 ;

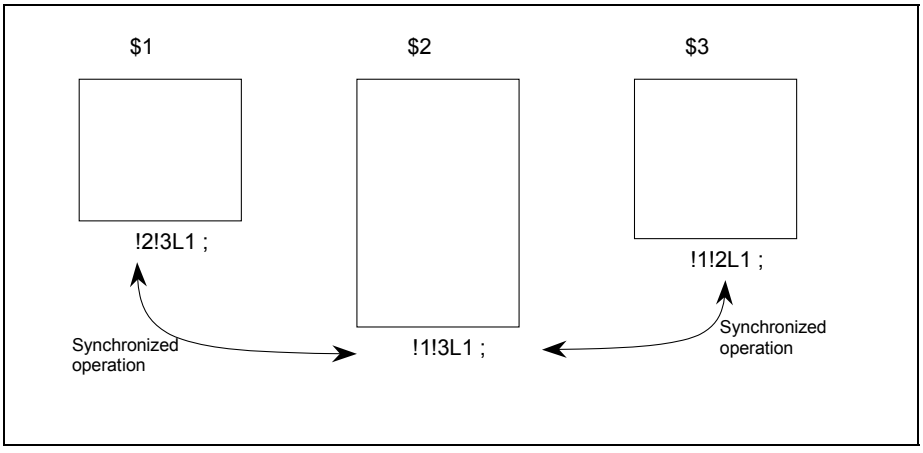
n : Part system number
1 : Synchronizing number 01 to 9999



(2) Command for synchronizing among three part systems

!n!m…L1 ;

n, m : Part system number n ≠ m
1 : Synchronizing number 01 to 9999



12.1.8.2 Start Point Designation Timing Synchronization

M system : O **L system : O**

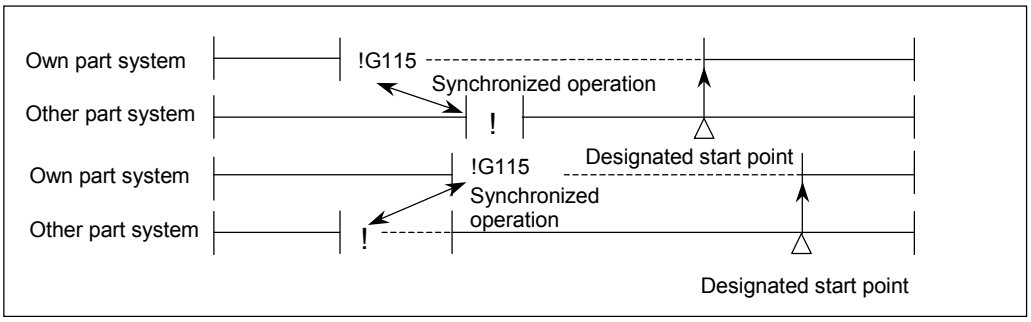
The synchronizing point can be placed in the middle of the block by designating the start point.

(1) Start point designation synchronization Type 1 (G115)

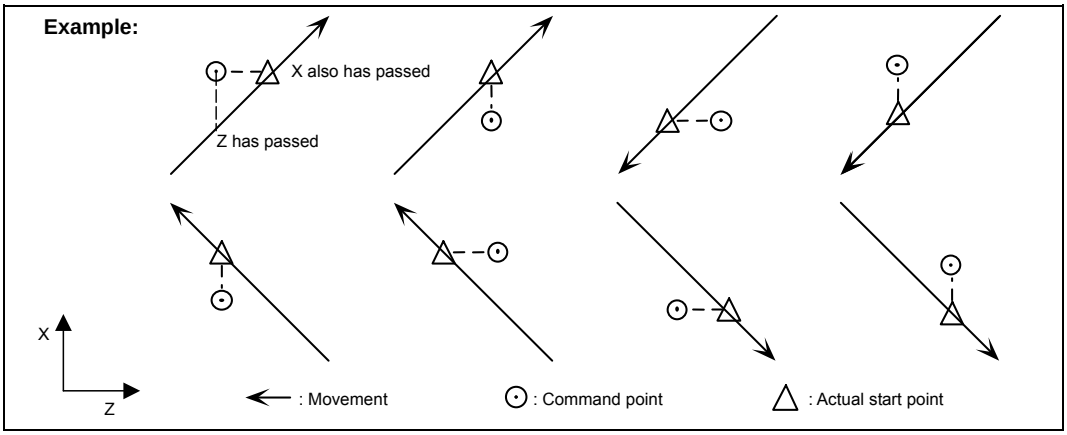
Command format

!Ll G115 X_ Z_ ;	
!Ll	: Synchronizing command
G115	: G command
X_, Z_	: Own start point (designate other part system's coordinate value)

(a) The other part system starts first when synchronizing is executed.
(b) The own part system waits for the other part system to move and reach the designated start point, and then starts.



(c) When the start point designated by G115 is not on the next block movement path of the other part system, the own part system starts once the other part system has reached all of the start point axis coordinates.

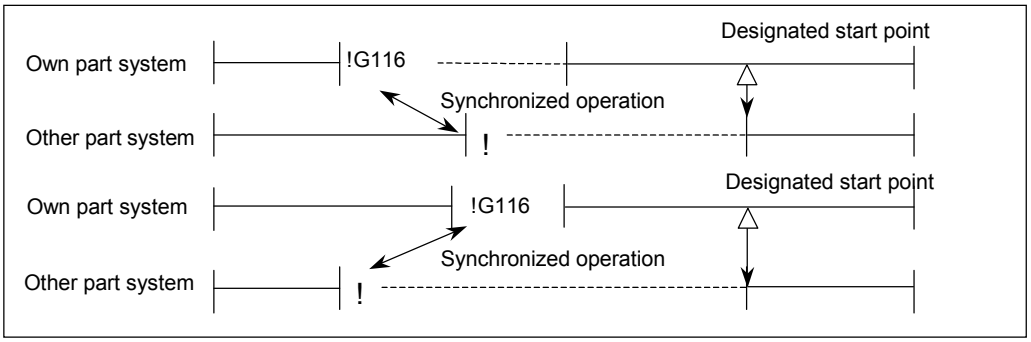


(2) Start point designation synchronization Type 2 (G116)

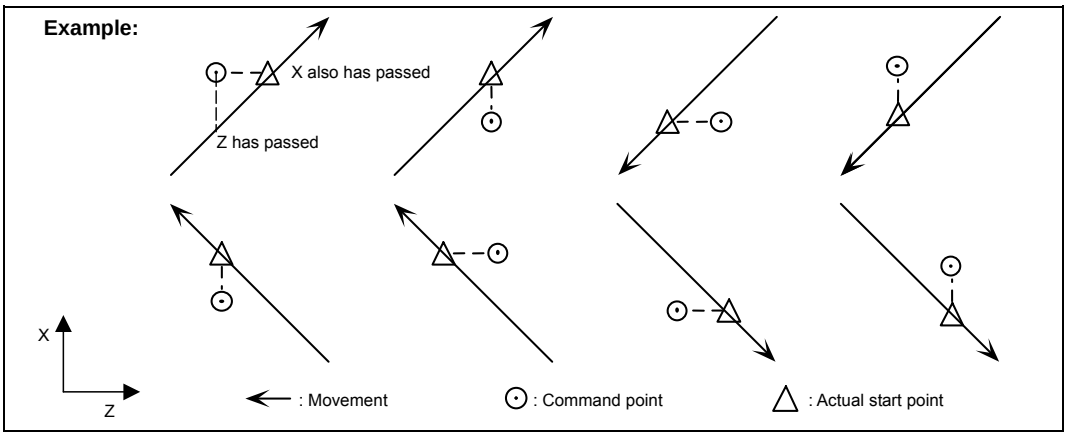
Command format

!LI G116 X_ Z_ ;	
!LI	: Synchronizing command
G116	: G command
X_, Z_	: Other start point (designate own part system's coordinate value)

(a) The own part system starts first when synchronizing is executed.
(b) The other part system waits for the own part system to move and reach the designated start point, and then starts.



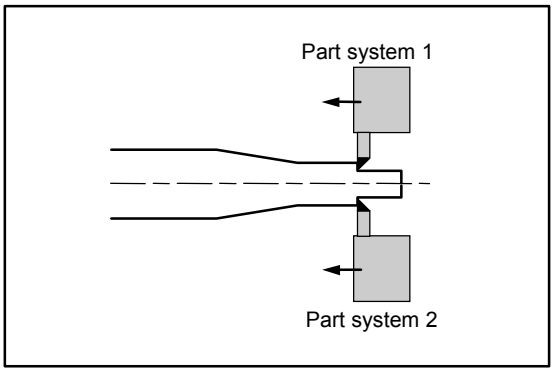
(c) When the start point designated by G116 is not on the next block movement path of the own part system, the other part system starts once the own part system has reached all of the start point axis coordinates.



12.1.8.6 Balance Cut

M system : - **L system : O**

When a workpiece that is relatively long and thin is machined on a lathe, deflection may result, making it impossible for the workpiece to be machined with any accuracy. In cases like this, the deflection can be minimized by holding tools simultaneously from both sides of the workpiece and using them in synchronization to machine the workpiece (balance cutting). This method has an additional advantage: since the workpiece is machined by two tools, the machining time is reduced.
The balance cutting function enables the movements of the tool rests belonging to part system 1 and part system 2 to be synchronized (at the block start timing) so that this kind of machining can easily be accomplished.



The command format is given below.

G14	Balance cut command OFF (modal)
G15	Balance cut command ON (modal)

G14 and G15 are modal commands. When the G15 command is assigned, the programmed operations of two part systems are synchronized (at the block start timing) for all blocks until the G14 command is assigned or until the modal information is cleared by the reset signal.

Part system 1 program

```
T0101;
G00 X_ Z_;
G15;
G01 Z_ F0.4;
    ⋮
```

Part system 2 program

```
T0102;
G00 X_ Z_;
G15;
G01 Z_ F0.4;
    ⋮
```

Whereas synchronization is possible only with the next block when using the code "!" of synchronization between part systems, the balance cutting function provides synchronization (at the block start timing) with multiple consecutive blocks.

12.1.8.8 2-part System Synchronous Thread Cutting

M system : - **L system : O**

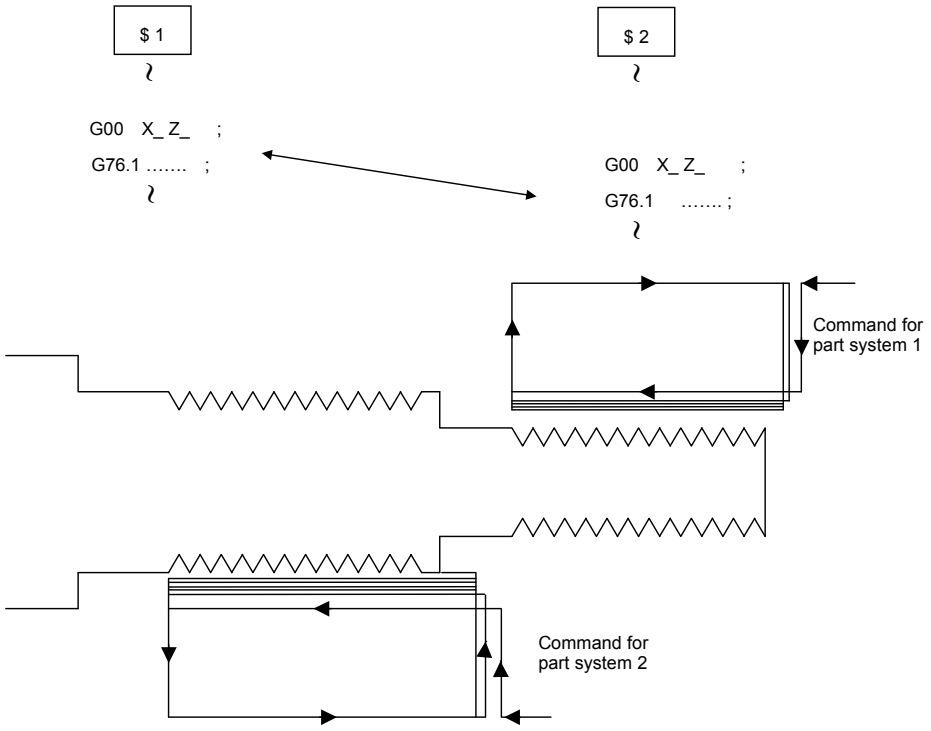
The 2-part system synchronous thread cutting cycle is the function which performs synchronous thread cutting for the same spindle using the part systems 1 and 2.
The 2-part system synchronous thread cutting cycle is "2-part system synchronous thread cutting cycle I" (G76.1) for synchronous thread cutting of two screws or "2-part system synchronous thread cutting cycle II" (G76.2) for thread cutting of one screw.

(1) 2-part system synchronous thread cutting cycle I

Command format

G76. 1 X/U_ Z/W_ Ri Pk Q∆d Fl ;	
X/U	: X axis end point coordinate of screw .... Designate the X coordinate of the end point at screw in an absolute or incremental value.
Z/W	: Z axis end point coordinate of screw .... Designate the Z coordinate of the end point at screw in an absolute or incremental value.
i	: Height constituent of taper at screw (radius value) ... When i is 0, a straight screw is generated.
k	: Screw thread height .... Designate the thread height in a positive radius value.
∆d	: Cut depth .... Designate the first cut depth in a positive radius value.
l	: Thread lead

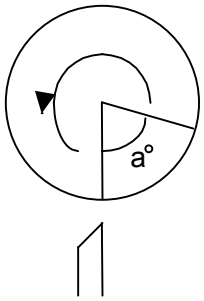
If G76.1 command is given in part system 1 or 2, a wait is made until G76.1 command is given in the other part system.
Once the G76.1 command exists in both part systems, the thread cutting cycle is started.



(2) 2-part system synchronous thread cutting cycle II

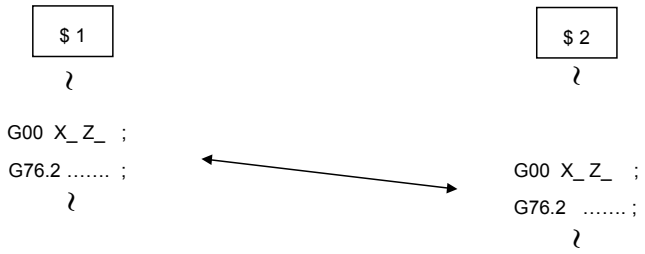
Command format

G76. 2 X/U_ Z/W_ Ri Pk Q∆d Aa Fl ; a : Thread cutting start shift angle

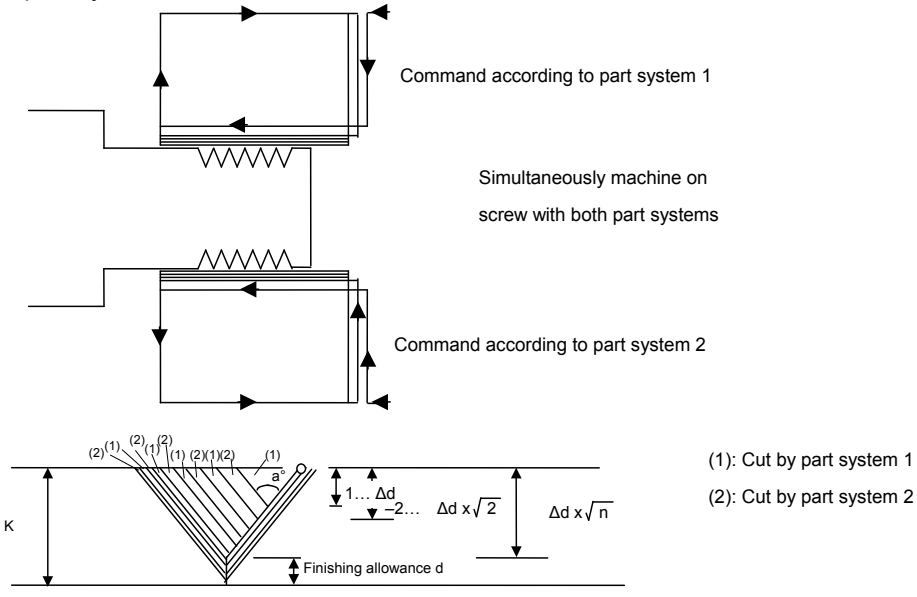


Thread cutting command waits for 1-revolution synchronizing signal of the spindle encoder and starts moving. The start point can be delayed by thread cutting start angle.

The address except A has the same meanings as those in 2-part system synchronous thread cutting cycle I.
If G76.2 command is given in part system 1 or 2, a wait is made until G76.2 command is given in the other part system. Once the G76.2 command exists in both part systems, the thread cutting cycle is started.



In the G76.2 cycle, the same screw is assumed to be cut, and it is cut deeply according to alternate cut depth in part systems 1 and 2.



12.1.9 Data Input by Program

12.1.9.1 Parameter Input by Program

M system : Δ **L system : Δ**

The parameters set from the setting and display unit can be changed using the machining programs.
The format used for the data setting is shown below.

G10 L70; Data setting command				
P parameter No.	S part system No.	A axis No.	H□ data ;	Bit parameter
P parameter No.	S part system No.	A axis No.	D data;	Numerical value parameter
P parameter No.	S part system No.	A axis No.	<character string>;	Character string parameter
P parameter No.	S part system No.	A axis 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 the same address is commanded more than twice, the last command will be valid.

(Note 2) The part system No. is set in the following manner: "1" for 1st part system, "2" for 2nd part system, and so forth.
If the address S is omitted, the part system of the executing program will be applied.
As for the parameters common to part systems, the command of part system No. will be ignored.

(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 the bit designation □ (0 to 7) and setting data (0 or 1).

(Note 5) Only the decimal number can be commanded with the address D.
The value that is smaller than the setting/display unit (#1003 iunit) will be rounded off.

(Note 6) Designate the character string with "," or "<>".
A program error (P33) will occur without either them.
Up to 31 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) If data with decimal point is commanded without decimal point, it is considered as decimal point valid.

12.1.9.2 Compensation Data Input by Program

M system : Δ **L system : Δ**

The value of the workpiece coordinate systems selected can be set or changed using program commands. The tool compensation amounts, that are set from the setting and display can be input by program commands.

(1) Workpiece coordinate system offset input

The value of the workpiece coordinate systems selected by the G54 to G59 commands can be set or changed by program commands.

G code			Function
G10	L2	P0	External workpiece coordinate system setting
G10	L2	P1	Workpiece coordinate system 1 setting (G54)
G10	L2	P2	Workpiece coordinate system 2 setting (G55)
G10	L2	P3	Workpiece coordinate system 3 setting (G56)
G10	L2	P4	Workpiece coordinate system 4 setting (G57)
G10	L2	P5	Workpiece coordinate system 5 setting (G58)
G10	L2	P6	Workpiece coordinate system 6 setting (G59)

The format for the workpiece coordinate system setting commands is shown below.

G10 L2 Pp1 Xx1 Yy1 Zz1 ;

G10 L2 : Parameter change command
Pp1 : Workpiece coordinate No.
Xx1, Yy1, Zz1 : Settings

(Note 1) L2 can be omitted. Omitting Pp1 results in a program error. [M system]

(2) Tool compensataion input

The tool compensataion 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]. The respective command format must be set by a parameter.

[M system]

G code	Function
G10 L10	Tool length shape compensataion amount
G10 L11	Tool length wear compensataion amount
G10 L12	Tool radius shape compensataion amount
G10 L13	Tool radius wear compensataion amount

The tool compensation input format is as follows.

G10 L10(L11/L12/L13) Pp1 Rr1 ;	
G10	L10(L11/L12/L13) : Command for setting compensataion amount
Pp1	: Compensataion No.
Rr1	: Compensataion amount

(Note 1) When L11(L12/L13) has been omitted, the tool length shape compensataion amount is set. Omitting Pp1 results in a program error.

[L system]

G code	Function
G10 L10	Tool length compensataion amount
G10 L11	Tool wear compensataion amount

The tool compensataion input format is as follows.

G10 L10(L11) Pp1 Xx1 Zz1 Rr1 Qq1 ;	
G10	L10(L11) : Command for setting compensataion amount
Pp1	: Compensataion No.
Xx1	: X axis compensataion amount
Zz1	: Z axis compensataion amount
Rr1	: Nose R compensation amount
Qq1	: Hypothetical tool nose point

12.1.10 Machining Modal

12.1.10.1 Tapping Mode

M system : ○ **L system : ○**

When tapping mode commands are issued, the CNC 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)
- High-accuracy control mode command (G61.1) [M system]

The machine is in the cutting mode status when its power is turned on.

12.1.10.2 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)
- High-accuracy control mode command (G61.1) [M system]

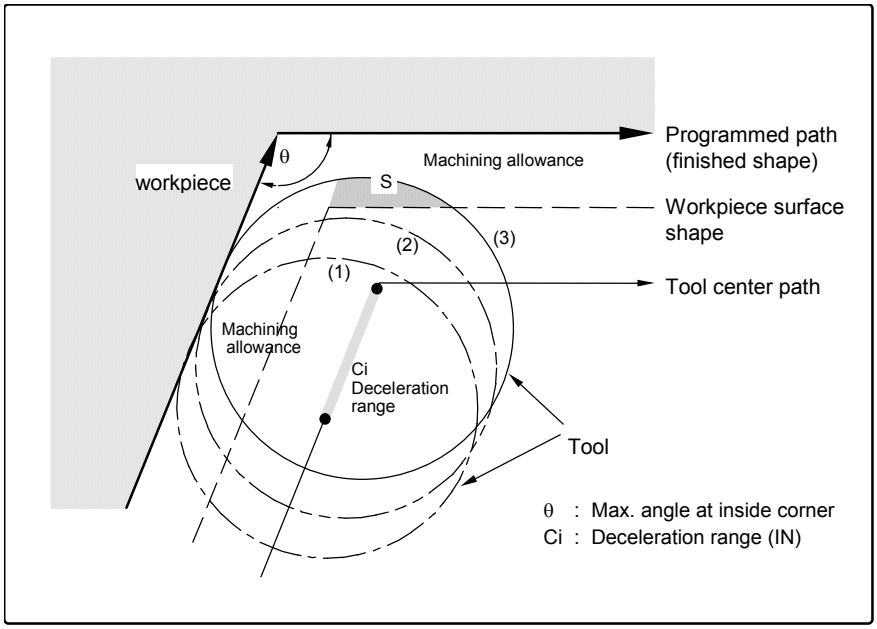
The machine is in the cutting mode status when its power is turned on.

12.2 Machining Accuracy Support Functions

12.2.1 Automatic Corner Override

M system : O **L system : O**

To prevent machining surface distortion due to the increase in the cutting load when cutting 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
G61.1 High-accuracy control mode [M system]



Operation

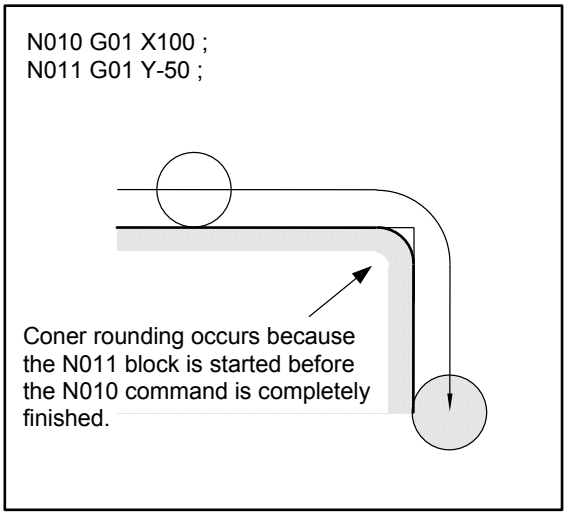
(a) When automatic corner override is not to be applied :
When the tool moves in the order of (1) → (2) → (3) in the figure above, the machining allowance at (3) is larger than that at (2) by an amount equivalent to the area of shaded section S and so the tool load increases.

(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.

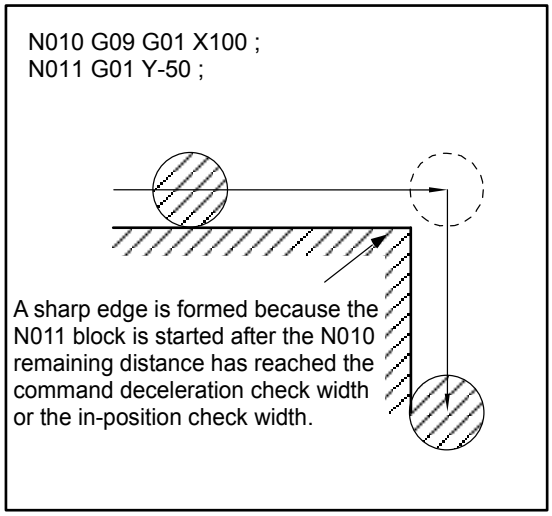
12.2.2 Deceleration Check

The deceleration check function leads the machine to decelerate and stop at the join between one block and another before executing the next block to alleviate the machine shock and to prevent the corner roundness that occurs when the feedrate of the control axis changes suddenly.

Without deceleration check



With deceleration check

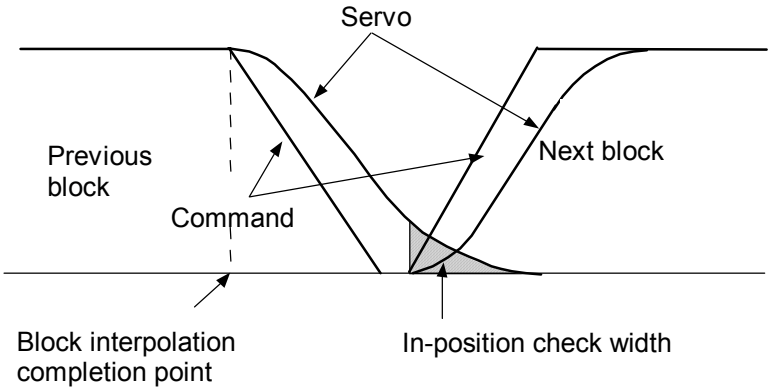


The conditions for executing deceleration check are described below.

(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).



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
G61.1 High-accuracy control mode [M system]

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 is 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 : O **L system : O**

This command is used to designate the in-position width, which is valid when a linear interpolation command is assigned, 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 X_ Z_ F_ ,I_;	
X_,Z_	: Linear interpolation coordinates of axes
F_	: Feed rate
,I_	: In-position width

This command is used to designate the in-position width, which is valid when a positioning command is assigned, from the machining program.

G00 X_ Z_ ,I_;	
X_,Z_	: Positioning coordinates of axes
,I_	: In-position width

In-position check operation

After it has been verified that the position error between the block in which the positioning command (G00: rapid traverse) is designated and 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.

12.3 High-speed and High-accuracy Functions [kBPM:k Block per Minute]

12.3.1 High-speed Machining Mode I (G5P1)

M system : Δ 16.8m/min L system : -

This function runs a machining program that approximates a free curves with fine segments at a high speed. This is effective for increasing the speed of machining dies with a free curve.

12.3.5 High-Accuracy Control 1 (G61.1/G08)

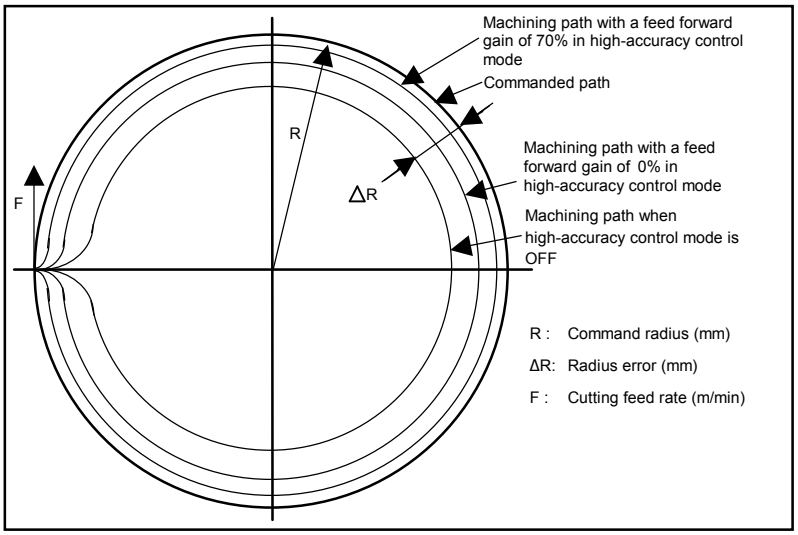
M system : Δ L system : -

This function controls the operation so the lag will be eliminated in control systems and servo systems. With this function, improved machining accuracy can be realized, especially during high-speed machining, and machining time can be reduced.

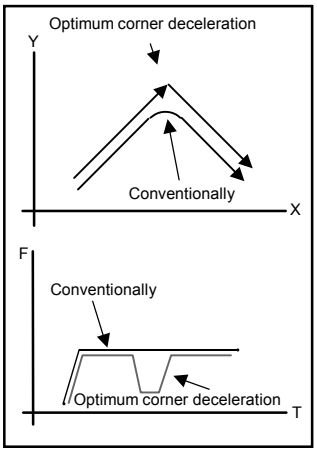
The high-accuracy control is commanded with ;

G61.1	High-accuracy control ON

Effects in G02/G03 circular interpolation



Neat machining of sharp corners without waste is realized with optimum linear acceleration/deceleration and corner judgement.



(1) Acceleration / deceleration before interpolation

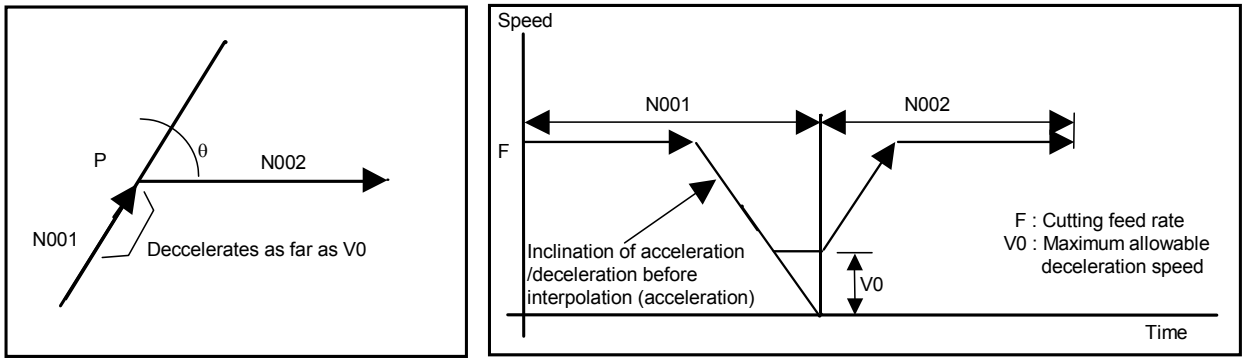
By accelerating /decelerating before interpolation, the machining shape error can be eliminated with smoothing, and a highly accurate path can be achieved.
With the arc commands, the radius reduction error can be significantly minimized.
Furthermore, since constant inclination acceleration/deceleration is performed, the time taken for positioning at microscopically small distances in the G00 command is reduced.

(Note 1) Whether acceleration/deceleration before interpolation in the rapid traverse command (G00) is to be performed always or not can be selected using a parameter setting independently from the high-accuracy control assignment.

(2) Optimum corner deceleration

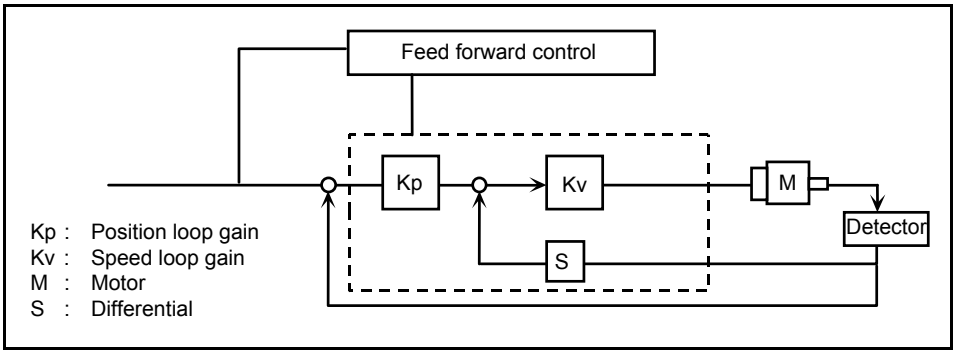
By determining the command vector in the machining program and thereby performing corner deceleration, it is possible to machine workpiece with a high-edge accuracy. The figure below shows the pattern of the deceleration speed at the corners.
(Optimum corner deceleration is a function of high-accuracy control mode.)

The speed change can be smoothed by the S-shape filter, the machine vibration can be suppressed, and the surface accuracy improved.
At the corner, the vector commanded in the machining program is automatically determined, and the speed is decelerated at the corner. A highly accurate edge can be machined by decelerating at the corner.



(3) Feed forward control

A stable servo control with an extremely small servo error can be realized using the feed forward control characteristic to this CNC system.



13. Machine Accuracy Compensation

13.1 Static Accuracy Compensation

13.1.1 Backlash Compensation

M system : O **L system : O**

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 input unit. 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).

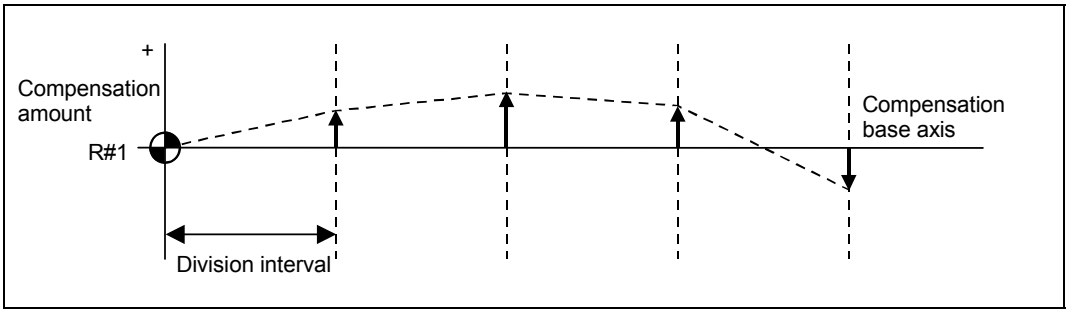
13.1.2 Memory-type Pitch Error Compensation

M system : Δ **L system : Δ**

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 (μm)
2. Number of compensation points : 1024
3. Compensation amount : -128 to 127 (output unit)
4. No. of compensated axes : 10 axes (including number of axes for relative position error compensation)

(1) The compensation position is set for the compensation axis whose reference point serves as the zero (0) point. Thus, memory-type pitch error compensation is not performed if return to reference point is not made for the compensation base axis or compensation execution axis 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) As shown in the figure above, highly individualized compensation control is exercised using the minimum output units 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 : Δ **L system : Δ**

Machine accuracy can be improved by compensating the relative error between machine axes, such as a production error or 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 (μm)
2. Number of compensation points	: 1024
3. Compensation amount	: -128 to 127 (output unit)
4. No. of compensated axe	: 10 axes (including number of axes for memory type pitch error compensation.)

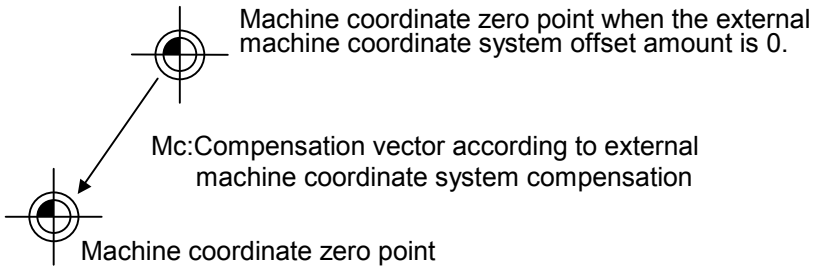
(1) The compensation position is set for the compensation axis whose reference point serves as the zero (0) point. Thus, memory-type relative position error compensation is not performed if return to reference point is not made for the compensation base axis or compensation execution axis after the controller 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 1024 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 : Δ **L system : Δ**

The coordinate system can be shifted by inputting a compensation amount from the PLC. This compensation amount will not appear on the counters (all counters including machine position). If the machine's displacement value caused by heat is input for example, this can be used for thermal displacement compensation.



13.1.5 Circular Error Radius Compensation

M system : Δ **L system : Δ**

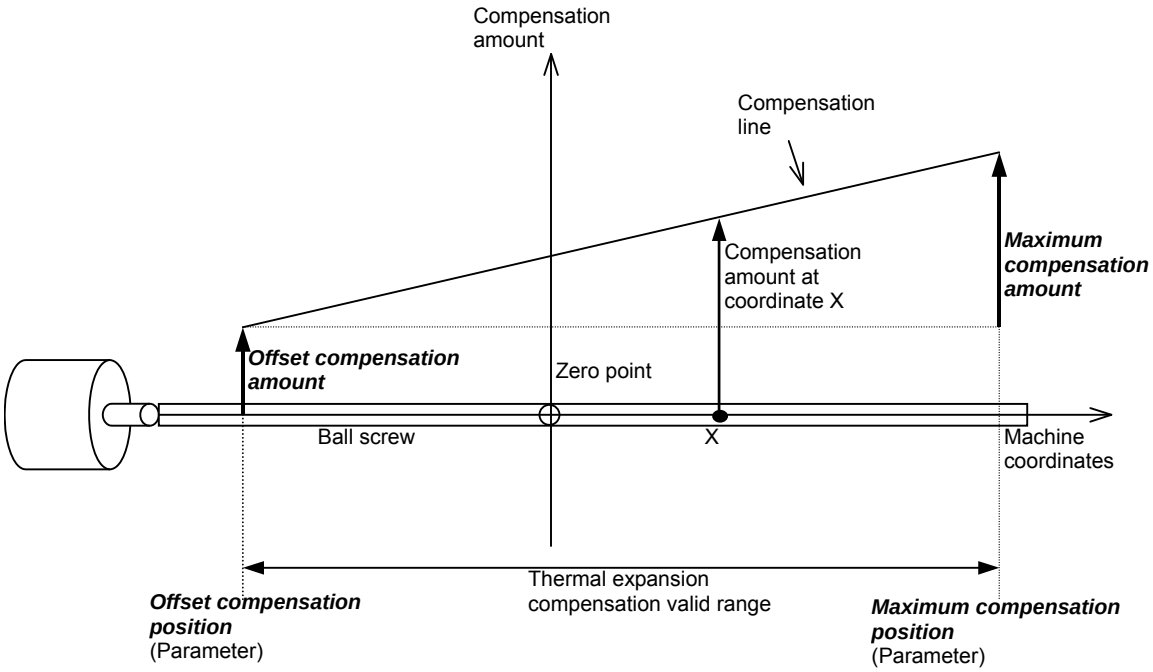
With commands designated during arc cutting, this function compensates for movement toward the inside of the arcs caused by a factor such as servo delay.

13.1.6 Ball Screw Thermal Expansion Compensation

M system : Δ **L system : Δ**

(1) Outline

The feed error caused by the thermal expansion of the ball screw is set from the PLC, and compensated.
The compensation amount depends on the offset compensation amount and maximum compensation amount.
The compensation amount based on the offset compensation amount is set as the maximum compensation amount.
The offset compensation amount and maximum compensation amount are set beforehand in the parameters.



(2) Compensation operation

The offset compensation position and maximum compensation position are connected with a straight line following the designated compensation amount, and the compensation amount to the current coordinates is obtained and compensated. The compensation amount changes immediately when the offset compensation amount or maximum compensation amount changes.
The thermal expansion compensation is valid only between the offset compensation amount and maximum compensation position, and is "0" outside of this range.
The compensation amount is not included in the coordinate value display.

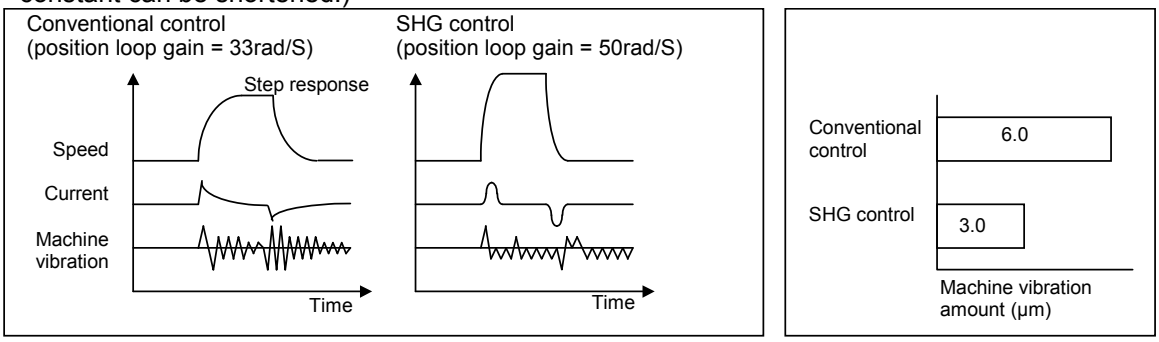
13.2 Dynamic Accuracy Compensation

13.2.1 Smooth High-gain (SHG) Control

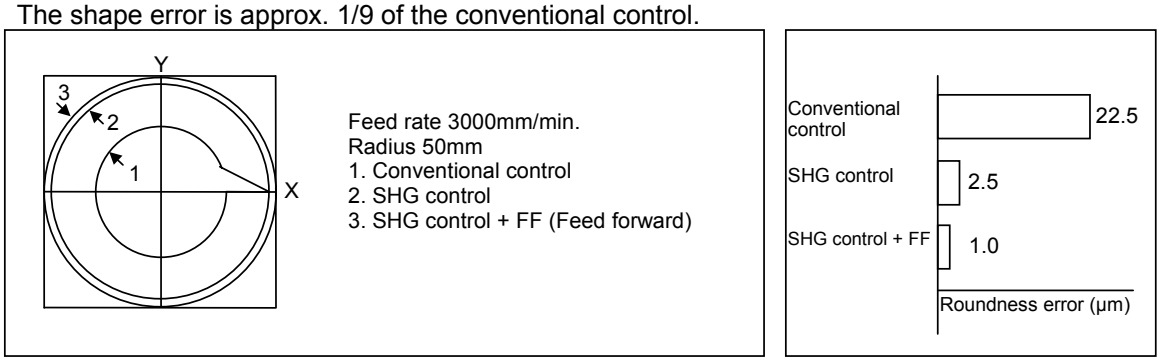
M system : O **L system : O**

This is a high-response and stable position control method using the servo system (MDS-□-V□/SVJ3). This SHG control realizes an approximately three-fold position loop gain compared to the conventional control method.
The features of the SHG control are as follows.

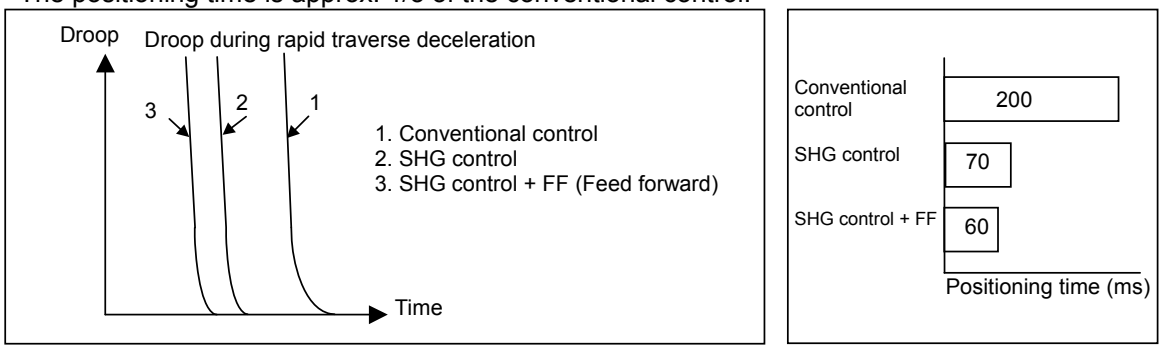
(1) The acceleration/deceleration becomes smoother, and the mechanical vibration can be suppressed (approx. 1/2) during acceleration/deceleration. (In other words, the acceleration/ deceleration time constant can be shortened.)



(2) The shape error is approx. 1/9 of the conventional control.



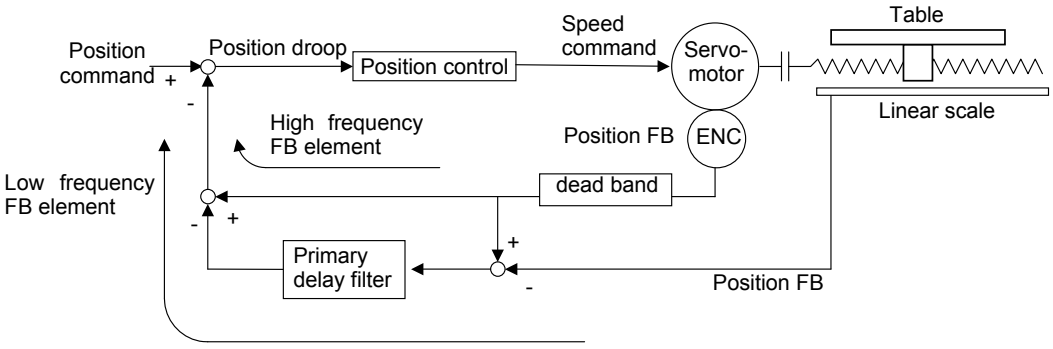
(3) The positioning time is approx. 1/3 of the conventional control.



13.2.2 Dual Feedback

M system : O **L system : O**

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.
A machine side detector (scale) is separately required.



Dual feedback control

The state will approach the semi-closed loop system as the primary delay filter's time constant increases, so the position loop gain limit will increase. Note that the limit of the position loop gain increased with the dual feedback function is the same as the position loop gain limit for a semi-closed system that does not use a machine side detector (scale, etc.). In addition, the positioning time will increase as the primary delay filter time constant increases.

13.2.3 Lost Motion Compensation

M system : O **L system : O**

This function compensates the error in protrusion shapes caused by lost motion at the arc quadrant changeover section during circular cutting.

14. Automation Support Functions

14.1 Measurement

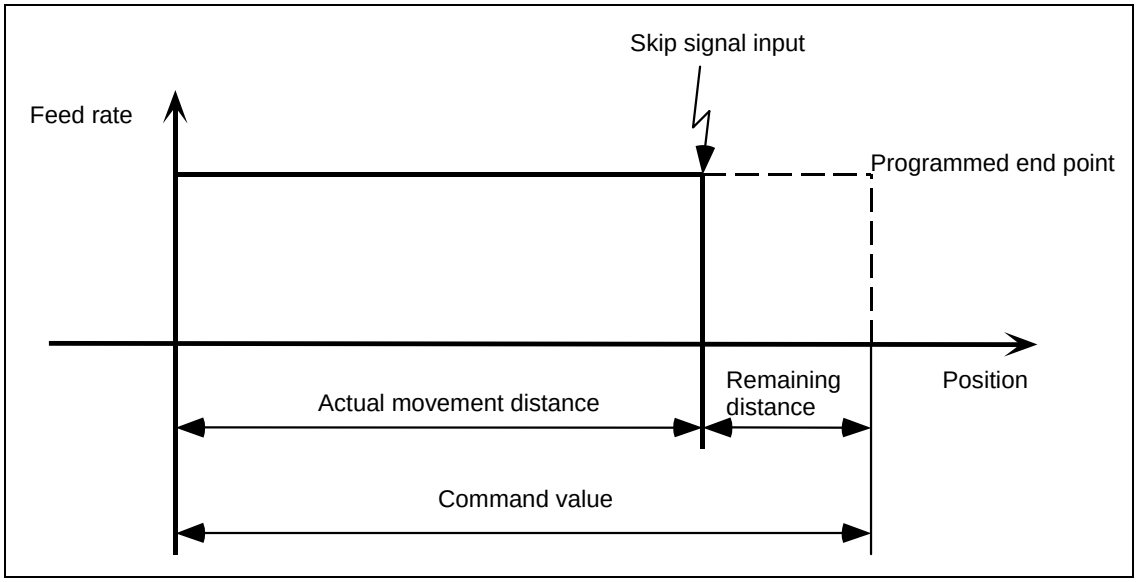
14.1.1 Skip

14.1.1.1 Skip

M system : Δ **L system : Δ**

When the external skip signal is input during linear interpolation using the G31 command, the machine feed is stopped immediately, the remaining distance is discarded and the commands in the next block are executed.

G31 Xx1 Yy1 Zz1 Ff1 ;	
G31	: Measurement command
Xx1, Yy1, Zz1	: Command values
Ff1	: Feed rate



When the G31 command is issued, acceleration/deceleration is accomplished in steps (time constant = 0).

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.

$$\delta \fallingdotseq \frac{F}{60} \times (Tp + t)$$

δ : Coasting distance (mm)
F : G31 rate (mm/min)
Tp : 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 : Δ **L system : Δ**

This function realizes skipping by designating a combination of skip signals for each skip command.

(1) G31.n method

This function realizes skipping by designating a combination of skip signals for each skip command (G31.1, G31.2, G31.3).
The combination of the skip signals 1, 2, 3 and 4 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	: Command format axis coordinate word and target coordinates
Ff1	: Feed rate (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 4. 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 and the next block executed under the skip conditions (to distinguish external skip signals 1 to 4) set with the parameters during the dwell command (G04).

G31 Xx1 Yy1 Zz1 Pp1 Ff1 ;	
G31	: Skip command
Xx1, Yy1, Zz1	: Command format axis coordinate word and target coordinates
Pp1	: Skip signal command
Ff1	: Feed rate (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 15. 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			
	4	3	2	1
1				○
2			○	
3			○	○
4		○		
5		○		○
6		○	○	
7		○	○	○
8	○			
:	:	:	:	:
13	○	○		○
14	○	○	○	
15	○	○	○	○

14.1.2 Automatic Tool Length Measurement

This function moves the tool in the direction of the tool measurement position by the commanded value between the measurement start position and measurement position. It stops the tool as soon as it contacts the sensor and calculates the difference between the coordinates when the tool has stopped and the command coordinates. It registers this difference as the tool length offset amount for that tool.

M system : Δ **L system : Δ**

(1) Automatic Tool Length Measurement (M system)

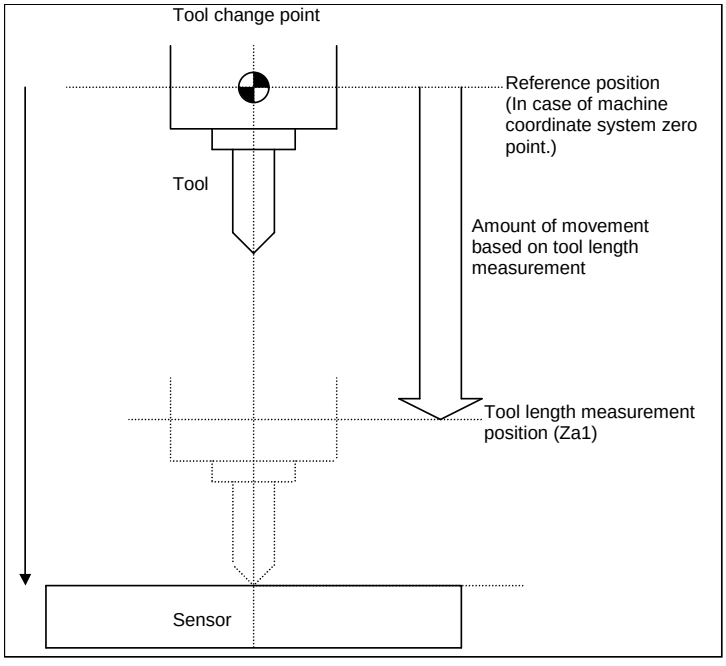
This function moves the tool in the direction of the tool measurement position by the commanded value between the measurement start position to the measurement position, it stops the tool as soon as it contacts the sensor and calculates the difference between the coordinates when the tool has stopped and commanded coordinates. It registers this difference as the tool length offset 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 Z_R_D_F_ ;

Z	: Measurement axis address and measurement position coordinate. ... X, Y, Z, α (where α is an optional axis)
R	: The distance between the point at which tool movement is to start at the measurement speed and the measurement position.
D	: The range in which the tool is to stop.
F	: The measurement rate.

When R_, D_ and F_ have been omitted, the values set in the parameters are used.



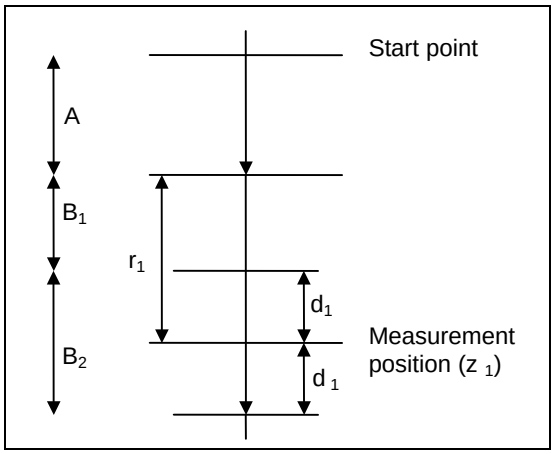
At this time, the tool length offset amount has a minus ("-") value.

Example of program

```
G28  Z0 ;
T01 ;
M06  T02 ;
G43  G00  Z0  H01 ;
G37  Z–300.  R10.D2.F10 ;
  .
  .
```

In this case, the distance (H01 = Za1 – z0) from the tool T01 tip to the top of the measurement sensor is calculated as the tool length offset amount which is then registered in the tool offset table.

(Note 1) The measurement position arrival signal (sensor signal) is also used as the skip signal.



Area A	: Moves with rapid traverse feed rate.
Areas B1, B2	: Moves with the measurement speed (f_1 or parameter setting)

If a sensor signal is input in area B_1, an error will occur.
If a sensor signal is not input in the area B_2, an error will occur.

(2) Automatic tool length measurement (L system)

This function moves the tool in the direction of the tool measurement position by the commanded value 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 coordinates when the tool has stopped and commanded coordinates. It registers this difference as the tool length offset 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 α_R_D_F_ ;

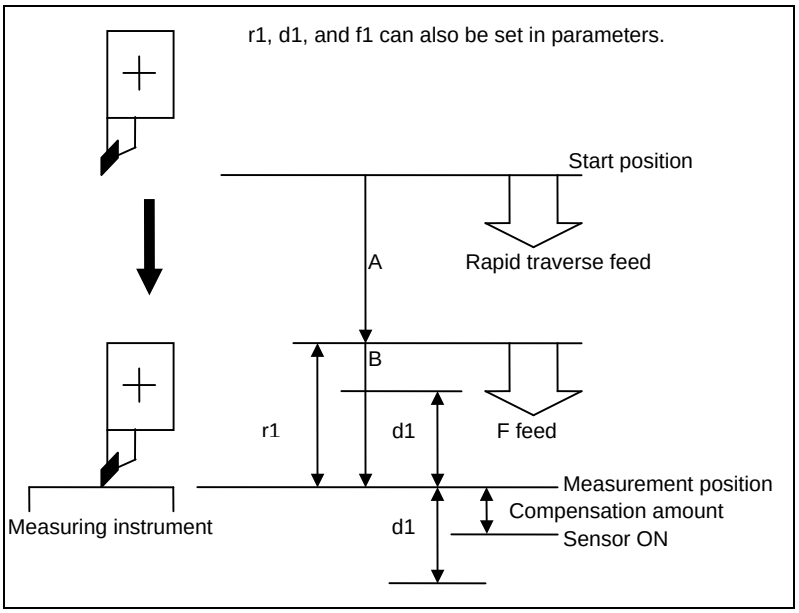
α : Measurement axis address and measurement position coordinate. ... X, Z

R : The distance between the point at which tool movement is to start at the measurement speed and the measurement position. (Always a radial value: incremental value)

D : The range in which the tool is to stop. (Always a radial value: incremental value)

F : The measurement rate.

When R_, D_ and F_ have been omitted, the values set in the parameters are used.



When the tool moves from the 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 d1, an error occurs.

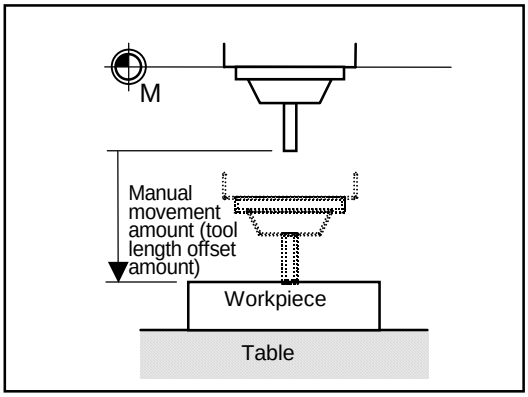
(Note 1) The measurement position arrival signal (sensor signal) is also used as the skip signal.
(Note 2) This is valid for the G code lists 2 and 3.

14.1.3 Manual Tool Length Measurement 1

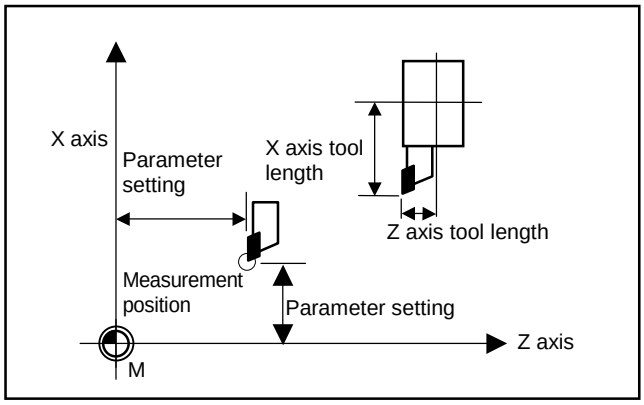
M system : Δ **L system : Δ**

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 point, 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 offset amount.



(2) Manual tool length measurement I
[L system]
A measurement position (machine coordinates) to match the tool nose on the machine is preset and the tool nose is set to the measurement position by manual feed, then the operation key is pressed, thereby automatically calculating the tool offset amount and setting it as the tool length offset amount.



Measurement method

(a) Preset the machine coordinates of the measurement position in a given parameter as the measurement basic value.
(b) Select a tool whose tool length offset amount is to be measured.
(c) Set the tool nose to the measurement position by manual feed.
(d) Press the input key. The tool length offset amount is calculated and displayed on the setting area.
Tool length offset amount = machine coordinates - measurement basic value
(e) Again press the input key to store the value in the memory as the tool length offset amount of the tool.

14.2 Tool Life Management

14.2.1 Tool Life Management

14.2.1.1 Tool Life Management I

M system : Δ **L system : Δ**

(1) M system

For the tool mounted on the spindle, 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 of up to 100 tools can be managed.

(2) L system

Tool life management is performed using the amount of time and frequency of use of a tool.
The life for up to 80 tools (tool numbers 1 to 80) can be managed.

(a) 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.
If the use time reaches the life time when a tool selection command is specified, an alarm is given.

(b) 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.
If the counter reaches the limit number when a tool selection command is specified, an alarm is given.

14.2.1.2 Tool Life Management II

M system : Δ **L system : Δ**

(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 a tool selection (T) command, 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.
- No. of groups: Max. 100 sets
- No. of tools in group: 100 tools (no limitation)

(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.
- No. of groups: Max. 40 sets (each part system)/ For 1 part system: 80 sets
- No. of tools in group: Max. 16 tools

14.2.2 Number of Tool Life Management Sets

The number of tools that can be managed for their lives are shown below. (These are fixed by the No. of part systems according to the model.)

14.2.2.1 80 sets

M system : - **L system : Δ**

Maximum of 80 tools for one part system and maximum of 40 tools for two or more part systems.

14.2.2.2 100 sets

M system : Δ **L system : -**

Up to 100 tools regardless of the number of part systems.

14.3 Others

14.3.1 Programmable Current Limitation

M system : ○ **L system : ○**

This function allows the current limit value of the servo axis to be changed to the 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 window function and 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 ;	
L14	: Current limit value setting (+ side/– side)
X	: Axis address
dn	: Current limit value 1% to 300%

(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 300%. Commands that exceed this range will cause a program error.
"P35 CMD VALUE OVER" will be displayed.
(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".

15. Safety and Maintenance

15.1 Safety Switches

15.1.1 Emergency Stop

M system : O **L system : O**

All operations are stopped by the emergency stop signal input, and at the same time, the drive section is shutoff to stop movement of the machine.
The servo ready signal is turned OFF.

15.1.2 Data Protection Key

M system : O **L system : O**

With the input from the user PLC, it is possible to prohibit the setting and deletion of parameters and the editing of programs from the 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, common variables, CNC ladder, R register data, C register data and T register data
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 CNC 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.

15.2.2 NC Alarm

M system : ○ **L system : ○**

The alarms which are output by the CNC 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 warning	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 with the register at the time when the error occurred on the screen if the system stops due to a system error.
Absolute position detection system alarm	An alarm in the absolute position detection system 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.

15.2.3 Operation Stop Cause

M system : ○ **L system : ○**

The stop cause of automatic operation is shown on the display.

15.2.4 Emergency Stop Cause

M system : ○ **L system : ○**

When the "EMG" (emergency stop) message is displayed in the operation status area of the setting and display unit, the cause of the emergency stop can be confirmed.

15.2.5 Thermal Detection

M system : ○ **L system : ○**

When overheating is detected in the control unit or the CNC CPU module, an alarm is displayed and the "temperature rise" signal is output at the same time. If the system is in auto run at the time, run is continued, but it cannot be started after reset or M02/M30 run ends. (It can be started after block stop or feed hold.)
When the temperature falls below the specified temperature, the alarm is released and the temperature rise signal is turned OFF.

15.2.6 Battery Alarm/Warning

M system : ○ **L system : ○**

When it is time for changing batteries, an alarm and warning are 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 : O **L system : O**

Limit switches and dogs are attached to the machine, and when 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

This function sets the prohibited area for the tool to enter.
The stored stroke limits I, II, IIB, IB and IC are handled as follows.

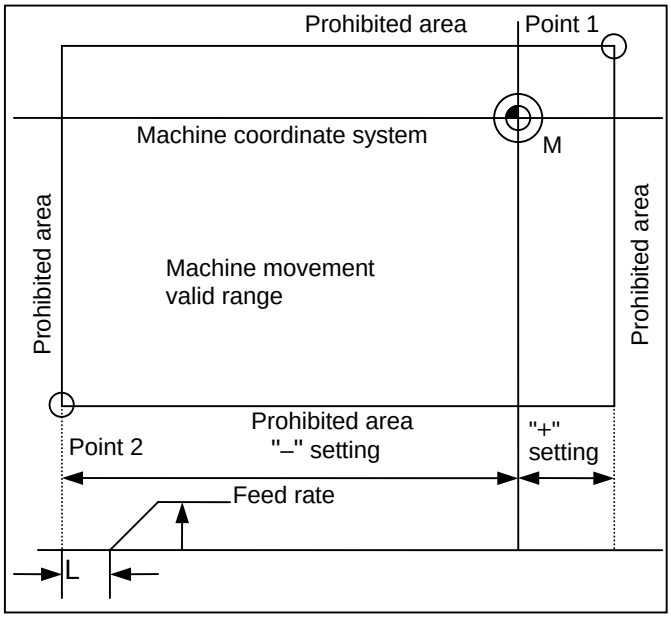
Type	Prohibited range	Explanation
I	Outside	• Set by the machine tool builder. • When used with II, the narrow range designated by the two types becomes the movement valid range. • Can be rewritten with window function.
II	Outside	• Set by the user. • The change or function of parameter can be turned OFF/ON with the program command. • Select II or IIB with the parameters. • Can be rewritten with window function.
IIB	Inside	
IB	Inside	• Set by the machine tool builder.
IC	Outside	• Set by the machine tool builder. • Can be rewritten with window function.

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 maker, 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 point return.
The stored stroke limit I function will be invalidated if the maximum and minimum values are set to the same data.

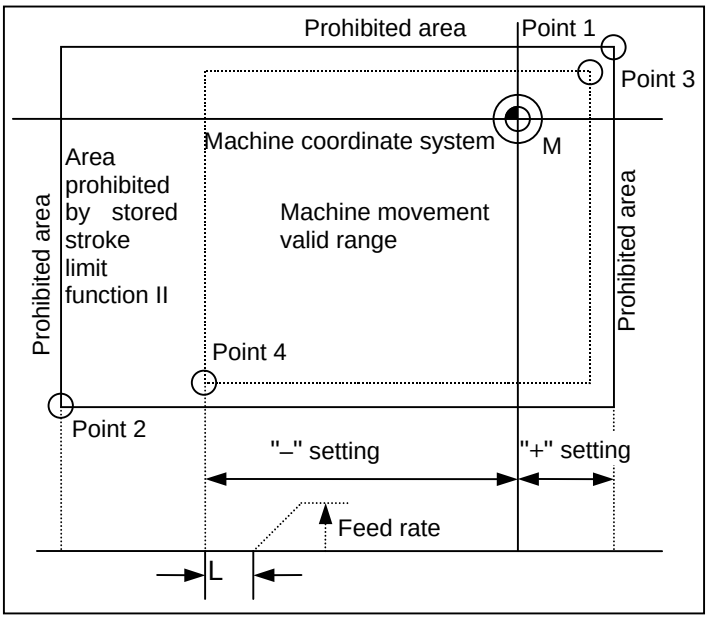


The values of points 1 and 2 are set using the coordinate values in the machine coordinate system.

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.

(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 values of points 3 and 4 are set with the coordinate values in the machine coordinate system.

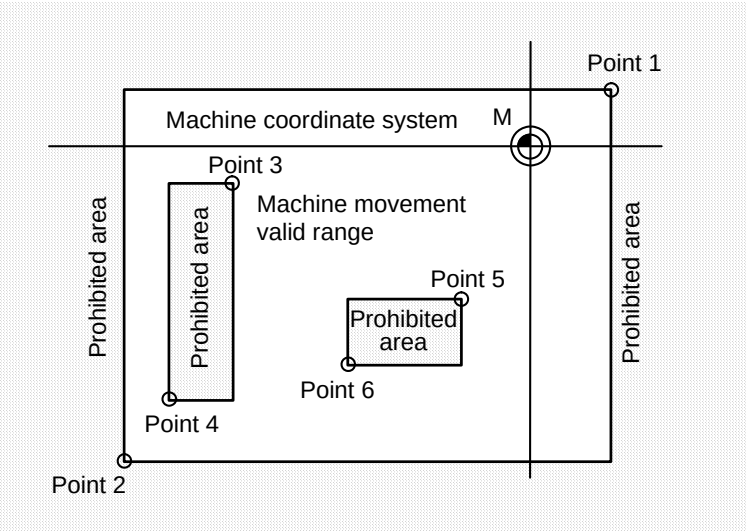
The area determined by points 1 and 2 is the prohibited area set with stored stroke limit I.

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.

15.3.2.2 Stored Stroke Limit IB

M system : Δ **L system : Δ**

Three areas where tool entry is prohibited can be set using the stored stroke limit I, stored stroke limit II, IIB and stored stroke limit IB functions.



The area determined by points 1 and 2 is the prohibited area set with stored stroke limit I.

The area determined by points 3 and 4 is the prohibited area set with stored stroke limit IIB.

The area determined by points 5 and 6 is the prohibited area set with stored stroke limit IB.

When an attempt is made to move the tool beyond the set range, an alarm is displayed, and the tool decelerates and stops. If the tool has entered into the prohibited area and an alarm has occurred, it is possible to move the tool only in the opposite direction to the direction in which the tool has just moved. This function is an option.

Precautions

- Bear in mind that the following will occur if the same data is set for the maximum and minimum value of the tool entry prohibited area:
  (1) When zero has been set for the maximum and minimum values, tool entry will be prohibited in the whole area.
  (2) If a value other than zero has been set for both the maximum and minimum values, it will be possible for the tool to move in the whole area.

15.3.2.3 Stored Stroke Limit IIB

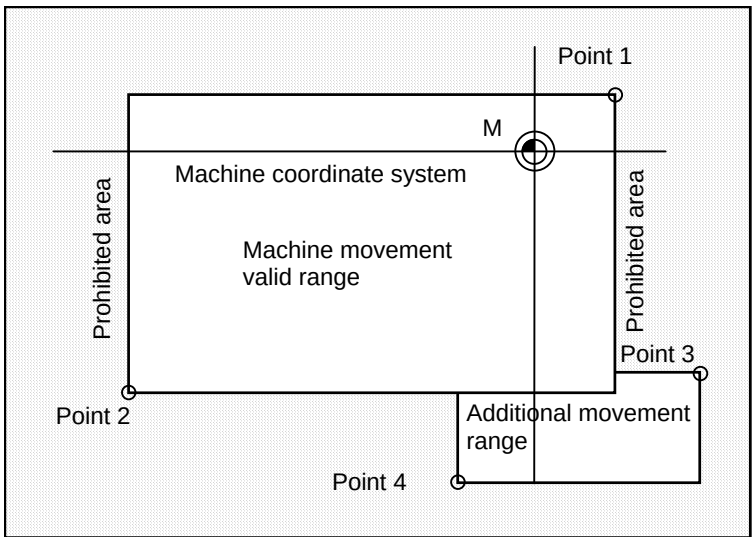
M system : Δ **L system : Δ**

A parameter is used to switch between this function and stored stroke limit II. With stored stroke limit IIB, the range inside the boundaries which have been set serves as the tool entry prohibited area.

15.3.2.4 Stored Stroke Limit IC

M system : Δ **L system : Δ**

The boundary is set for each axis with the parameters. The inside of the set boundary is the additional movement range.
This cannot be used with soft limit IB.



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.

15.3.4 Chuck/Tailstock Barrier Check

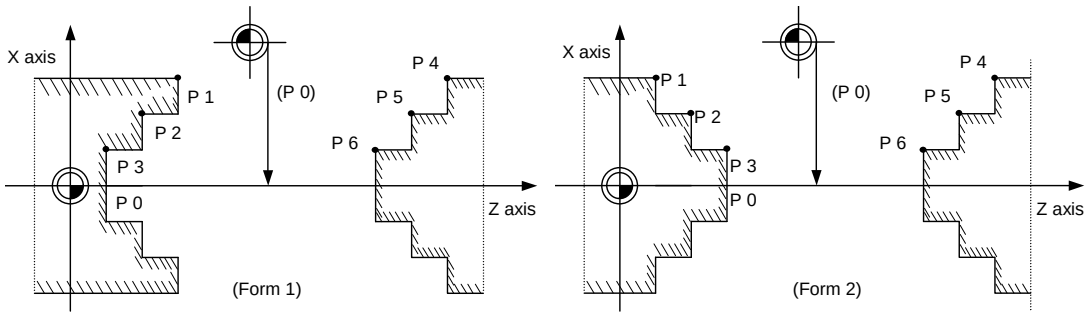
M system : - **L system : O**

By limiting the tool nose point movement range, this function prevents the tool from colliding with the chuck or tail stock 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 point return.
(4) The chuck barrier/tail stock barrier can be set independently for part system 1 and part system 2.
(5) Chuck barrier/tail stock barrier setting



The chuck barrier and tail stock 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 tail stock barrier. The X axis is set with the coordinate value (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 tail stock barrier's basic X coordinates, and the workpiece center coordinate in the basic machine coordinate system is set.
The barrier area is assumed to be symmetrical for the Z axis, and if the X axis coordinate of barrier point P_ is minus, the sign is inverted to plus and the coordinate is converted for a check.
Set the absolute values of the X axis coordinates of the barrier points as shown below:
P1 >= P2 >= P3, P4 >= P5 >= P6
(However, this need not apply to the Z axis coordinates.)

15.3.5 Interlock

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.
(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.

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 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. External deceleration speed can be set commonly for axes of each part system, or it can be set for each axis. The choice of which setting to use can be set with a parameter. 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.

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 protective doors while a machine is actually moving is prohibited.
When the door open signal is input from the PLC, this function first decelerates, stops all the control axes, establishes the ready OFF status, and then shuts off the drive power inside the servo drive units 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.
  1. 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.
  2. 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 servo 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.

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 protective doors while a machine is actually moving is prohibited.
When the door open signal is input from the PLC, this function first decelerates, stops all the control axes, establishes the ready OFF status, and then shuts off the drive power inside the servo drive units 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 servo 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

(Note 1) Concerning the handling of an analog spindle
The signals described in this section are valid in a system with serial 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

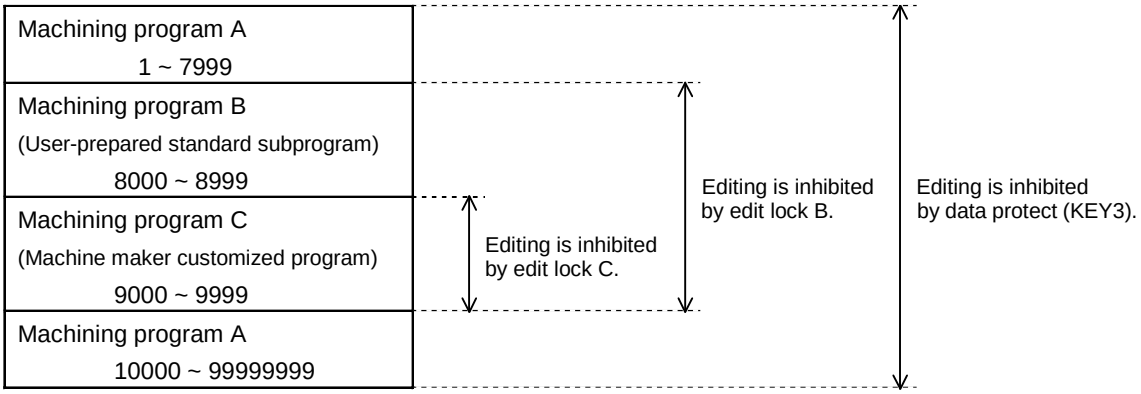
M system : O **L system : O**

This function is used to prohibit changing the set-up parameter.

15.3.11 Program Protection (Edit Lock B, C)

M system : O **L system : O**

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 protection.



15.3.12 Program Display Lock

M system : O **L system : O**

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.

The program details are not displayed in the prohibited state, but the program number and sequence number will be displayed.

15.3.13 Safety Observation

M system : Δ **L system : Δ**

This function is composed of the following three functions.

[Dual safety circuit function]
PLC CPU and NC CPU separately control the Input/Output signal of the dual signal unit.
The state of the disagreement of the Input/Output signal of each CPU is observed by the dual-signal comparison. When an error is detected during observation, the main power for the drive is shut.

[Dual emergency stop function]
PLC CPU, NC CPU and drive CPU separately observe the input of emergency stop.
The main power for the drive can be shut by controlling the contactor from PLC, NC, and drive CPU respectively when the emergency stops.

[Dual speed monitor function]
CNC CPU and drive CPU separately observe the following.

- Observe the open and close state signal of the safety door detected with a different circuit
- Observe that the command speed should not exceed the speed set by the parameter (safety speed).
- Observe that the motor rotation speed should not exceed the rotation speed set by parameter (safety rotation speed).

When an error is detected during observation, the main power for the drive is shut.

15.4 Maintenance and Troubleshooting

15.4.1 Operation history

M system : ○ **L system : ○**

This is a maintenance function which is useful for tracing the history and CNC operation information, and analyzing trouble, etc. This information can be output as screen displays or as files.

(1) Screen display showing operation history and event occurrence times
The times/dates (year/month/day and hour/minute/second) and messages are displayed as the operation history data. The key histories, alarm histories and input/output signal change histories are displayed as the messages.
The part system information is displayed as the alarm histories.
For instance, "$1" denotes the first part system, and "$2" the second part system.
The history data containing the most recent operation history and event occurrence times (2,068 sets) are displayed on the "Operation history" screen. The most recent history data appears at the top of the screen, and the older data is displayed in sequence below.

(2) Outputting the data in the operation history memory
Information on the alarms occurring during NC operation and stop codes, signal information on the changes in the PLC interface input signals and the key histories can be output.

15.4.2 Data Sampling

M system : ○ **L system : ○**

The data sampling function can sample the CNC internal data (speed output from the CNC to the drive unit and feedback data from the drive unit, etc.) and output it as text data.

15.4.3 NC Data Backup

M system : ○ **L system : ○**

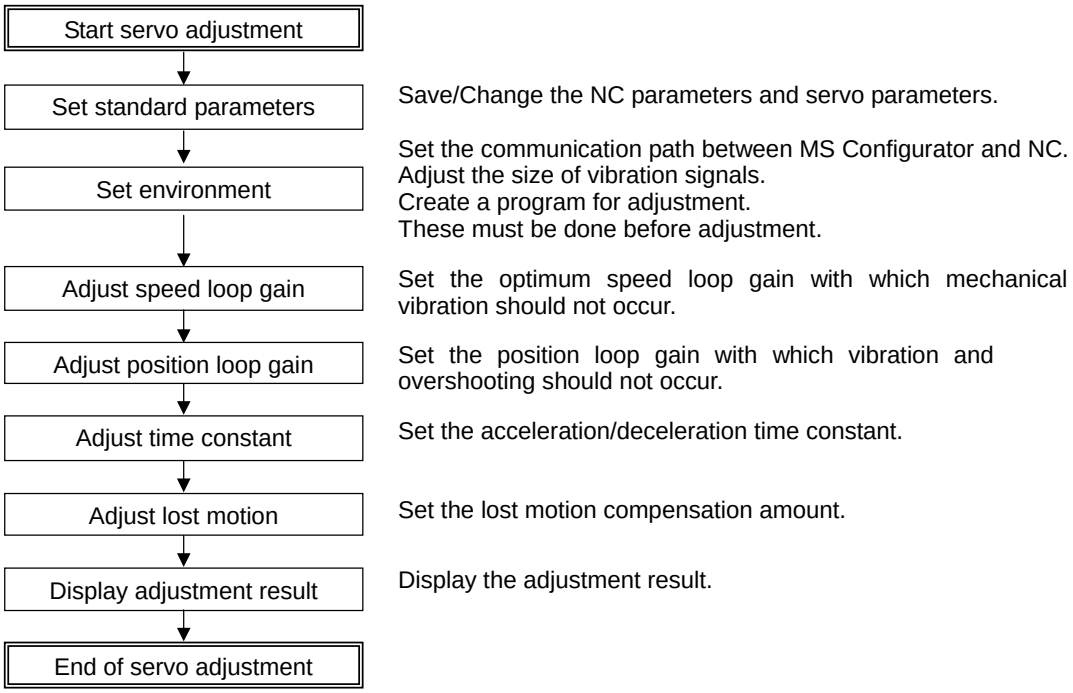
This function serves to back-up the parameters and other data of the CNC control unit.
The data can also be restored.

15.4.5 Servo Automatic Tuning (MS Configurator)

M system : O **L system : O**

With this function, the servo parameters can be automatically adjusted by connecting the CNC and MS Configurator, which is an application that runs on a regular personal computer.
MS Configurator measures and analyzes the machine characteristics to automatically adjust the servo parameters while having the motor run by test NC programs or vibration signals.

The servo is adjusted with the MS Configurator according to the following flow.



MS Configurator supports the following servo parameter automatic adjustment function and data measurement function.

(1) Environment setting

Function	Details
Communication path setting	Sets the communication path with NC.
Vibration signal setting	Adjusts the size of vibration signals.
Parameter setting	Saves and changes the servo parameters.
Program creation	Creates program for adjustment.

(2) Automatic adjustment function

Function	Details
Speed loop gain adjustment	Automatically adjusts the speed loop gain.
Position loop gain adjustment	Automatically adjusts the position loop gain.
Time constant adjustment	Automatically adjusts the acceleration/deceleration time.
Lost motion adjustment	Automatically adjusts the quadrant protrusion amount of the designated axis.
Batch adjustment	Automatically adjusts the above 4 items.

15.4.102 Backup

M system : ○ **L system : ○**

This function saves (backs up) the screen data and each controller (PLC, CNC) data to a GOT CF card. It also reloads (restores) that data to each device.
If this function is used, the backup is unnecessary for the MONITOR screen and each controller, and work improves.

16. Drive System

CNC dedicated products are used as drive units, spindle motors, and servo motors.
Refer to the following manuals for details on the servo and spindle system.
MDS-D/DH Series Specifications Manual (IB-1500875)
MDS-D-SVJ3/SPJ3 Series Specifications Manual (IB-1500158)
MDS-DM Series Specifications Manual (IB-1500875)

16.1 Servo/Spindle

16.1.1 Servo Drive Unit

16.1.1.1 MDS-D-V1/D-V2 (200V)

(1) Servo motor : HF□□-A48 (260 kp/rev)

M system : □ **L system : □**

(2) Servo motor : HF-KP□□JW04 (260 kp/rev)

M system : □ **L system : □**

(3) Servo motor : HF-KP13□J

M system : □ **L system : □**

16.1.1.2 MDS-DH-V1/DH-V2 (400V)

Servo motor : HF-H□□-A48 (260 kp/rev)

M system : □ **L system : □**

16.1.1.3 MDS-D-SVJ3 (200V)

(1) Servo motor: HF**-A48 (260kp/rev)

M system : □ **L system : □**

(2) Servo motor: HF-KP**JW04 (260kp/rev)

M system : □ **L system : □**

(3) Servo motor : HF-KP13□J

M system : □ **L system : □**

16.1.1.4 MDS-DM-V3 (200V)

(1) Servo motor: HF□□-A48(260kp/rev)

M system : □ **L system : □**

(2) Servo motor: HF-KP□□JW04-S6(260kp/rev)

M system : □ **L system : □**

(3) Servo motor : HF-KP13□J

M system : □ **L system : □**

16.1.2 Spindle Drive Unit

16.1.2.1 MDS-D-SP/D-SP2 (200V)

M system : ☐ **L system : ☐**

16.1.2.2 MDS-DH-SP (400V)

M system : ☐ **L system : ☐**

16.1.2.3 MDS-D-SPJ3 (200V)

M system : ☐ **L system : ☐**

16.1.3 Multi-hybrid Drive Unit

16.1.3.1 MDS-DM-SPV2/SPV3 (200V)

M system : ☐ **L system : ☐**

16.1.4 Power Supply

16.1.4.1 Power Supply : MDS-D-CV (200V)

M system : ☐ **L system : ☐**

16.1.4.2 Power Supply : MDS-DH-CV (400V)

M system : ☐ **L system : ☐**

17. Machine Support Functions

17.1 PLC

17.1.2 PLC Functions

17.1.2.1 Built-in PLC Basic Function

M system : Δ (MELSEC) L system : Δ (MELSEC)

As the PLC function of MITSUBISHI CNC C70, the PLC CPU of MITSUBISHI Programmable Controller MELSEC is used.
Select a PLC suitable for the control scale and the performance from several kinds of PLCs and use it.
Refer to the material of MITSUBISHI Programmable Controller MELSEC which can be used with MITSUBISHI CNC C70 for details.

17.1.2.2 NC Exclusive Instruction

M system : Δ (MELSEC) L system : Δ (MELSEC)

NC exclusive instructions are not MELSEC standard instructions. They are directly related to the memory in the CNC, and convenient for using a CNC. They can be programmed with the MELSEC programming tool GX Developer as other standard instructions.

NC exclusive instructions include:

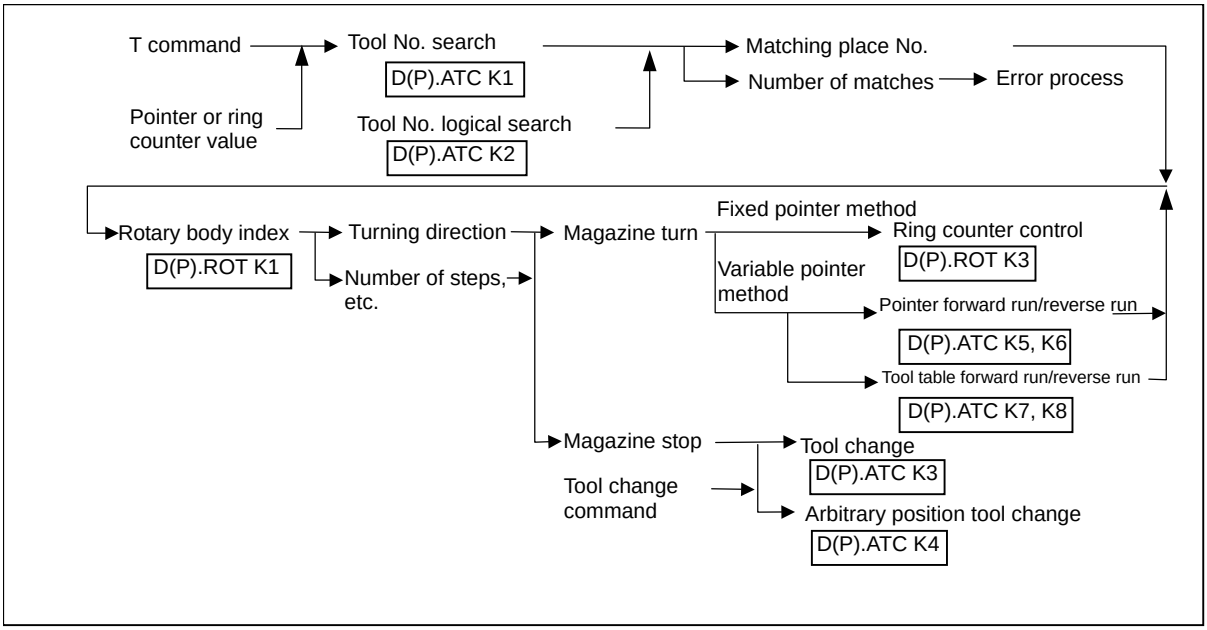
(1) ATC exclusive instruction (D(P).ATC)
This is an instruction to function ATC, or magazine index control, tool exchange with arm, etc.
ATC exclusive instructions are as follows.

- Tool No. search
- Tool change
- Tool table forward/reverse run
- Pointer (which indicates magazine index position) forward/reverse run
- Tool data read/write

(2) Rotary body control instruction (D(P).ROT)
This is an instruction to determine the rotary body's target position or rotation direction, or to function as a ring counter. This is used when calculating the rotation direction or number of index steps of the magazine or turret based on the output data figured from ATC exclusive instruction tool No. search processing, or when controlling the rotary body position.

Using the ATC and ROT instructions

The order for using the D(P).ATC and D(P).ROT instructions when T is commanded or tool exchange is commanded is shown below.



17.1.2.3 Built-in PLC Processing Mode

M system : ○ L system : ○

This function is used when executing safety observation of significant signals using a dual signal module. Refer to the documents of safety observation function for details.

17.1.3 PLC Support Functions

17.1.3.6 Multi-ladder Program Register and Execution

M system : Δ (MELSEC) L system : Δ (MELSEC)

Two or more PLC programs can be stored and executed.

17.1.3.7 Ladder Program Writing during RUN

M system : Δ (MELSEC) L system : Δ (MELSEC)

Ladder program can be written while CPU unit is running.

17.1.3.8 PLC Protection

M system : Δ (MELSEC) L system : Δ (MELSEC)

Read and writing of PLC program and device comments can be prohibited.

17.1.4 Built-in PLC Capacity

M system : □ 30k/40k/60k/130k/260k L system : □ 30k/40k/60k/130k/260k

Parameters, intelligent function unit parameters, programs, device comments, and system area set by user can be stored in a program memory.

17.1.5 Machine Contact Input/Output I/F

M system : Δ (MELSEC) L system : Δ (MELSEC)

The device is selected from the I/O unit of MITSUBISHI Programmable Controller MELSEC Q Series. Follow the manual of the I/O unit about the method of handling.

17.1.6 Ladder Monitor

M system : ○ L system : ○

This function enables the operating status of the sequence circuit to be checked on a MITSUBISHI Graphic Operation Terminal (GOT).
The monitor functions include the following.
(1) Circuit monitoring
(2) Batch monitor
(3) Entry monitoring

17.1.7 PLC Development

17.1.7.1 On-board development

M system : ○ **L system : ○**

This function enables to monitor and edit PLC circuit on the MITSUBISHI Graphic Operation Terminal (GOT) screen by using GOT's ladder edit function. For the On-board development of GOT, refer to the GOT materials.

17.1.7.2 MELSEC Development Tool (GX Developer)

M system : ○ **L system : ○**

This function enables the data of the MELSEC CPU PLC programs to be developed and debugged using GX Developer installed in a personal computer with Windows.
Many and varied functions of the GX Developer make it possible to reduce the PLC data development and debugging time.

17.1.7.3 MELSEC Development Tool (GX Simulator)

M system : ○ **L system : ○**

This function enables to run a simulation of PLC CPU on a personal computer with Windows. Debugging of the PLC program is also possible.

17.1.9 GOT Connection

For connecting a MITSUBISHI Graphic Operation Terminal (GOT), refer to the GOT materials.
Only when GOT has been bus-connected with the DISPLAY interface of NC CPU or the basic base unit, the CNC exclusive use screen (CNC monitor function) can be displayed.
The size of GOT corresponds to SVGA and XGA.

17.1.9.1 CPU Direct Connection (RS-422/RS-232C)

M system : Δ (MELSEC) **L system : Δ (MELSEC)**

17.1.9.2 CC-Link Connection (Remote device)

M system : Δ (MELSEC) **L system : Δ (MELSEC)**

17.1.9.3 CC-Link Connection (Intelligent terminal)

M system : Δ (MELSEC) **L system : Δ (MELSEC)**

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.

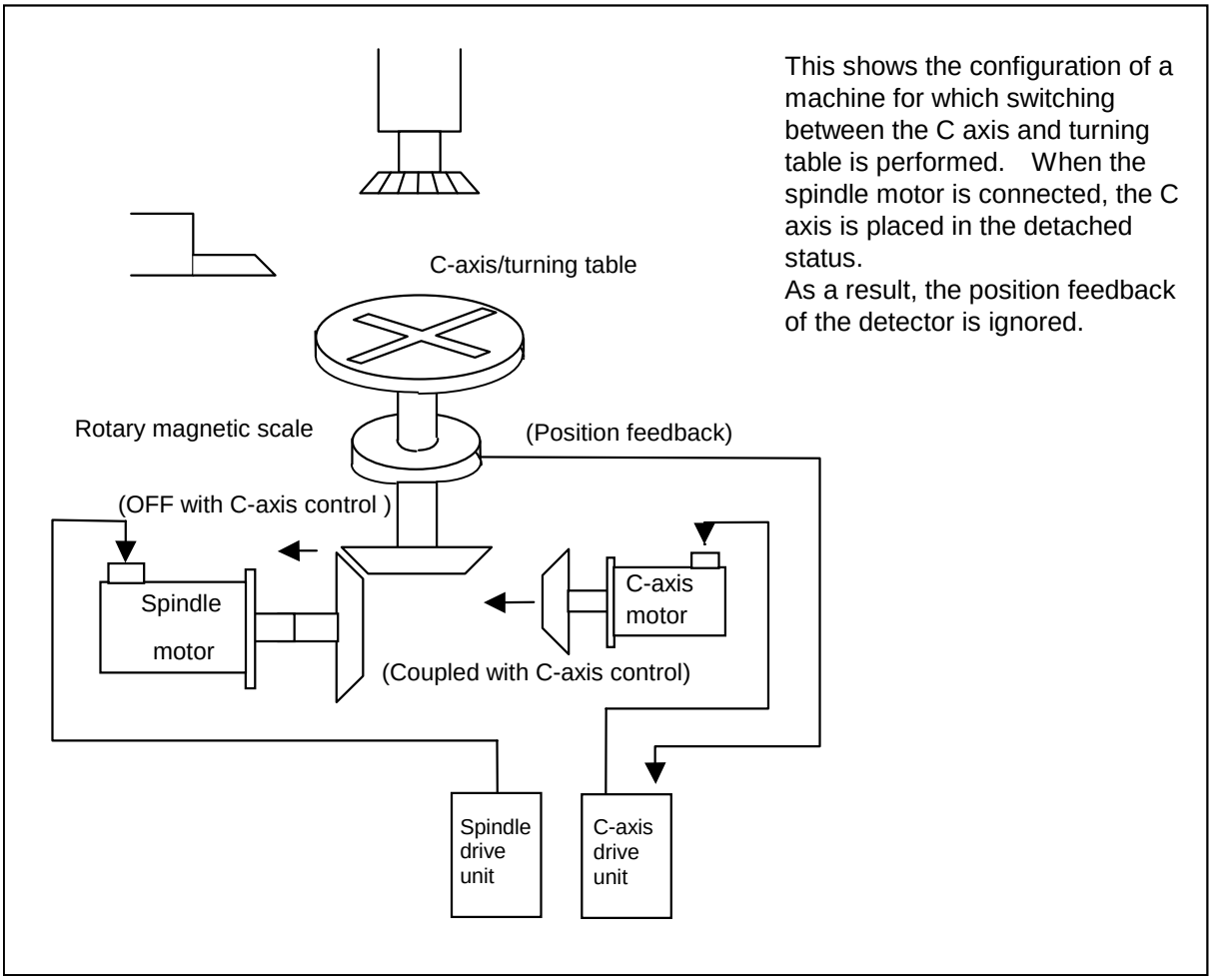
Mechanical handle

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 : Δ **L system : Δ**

This function enables the control axis to be released 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.

```
POSITION

   X    1 2 3 . 4 5 6
   Z        0 . 0 0 0 #1
   C    3 4 5 . 6 7 8 ><
```

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.

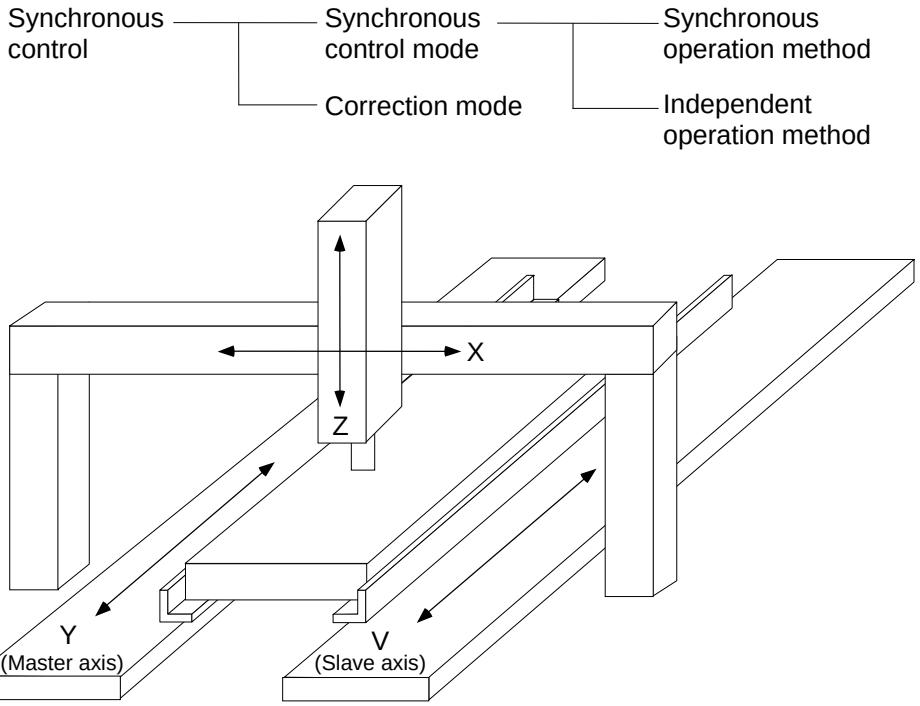
(Note) Axis detach can be executed even for the absolute position detection specifications axis, but when the axis is reinstalled, the zero point must be set.

17.2.3 Synchronous Control

M system : Δ **L system : -**

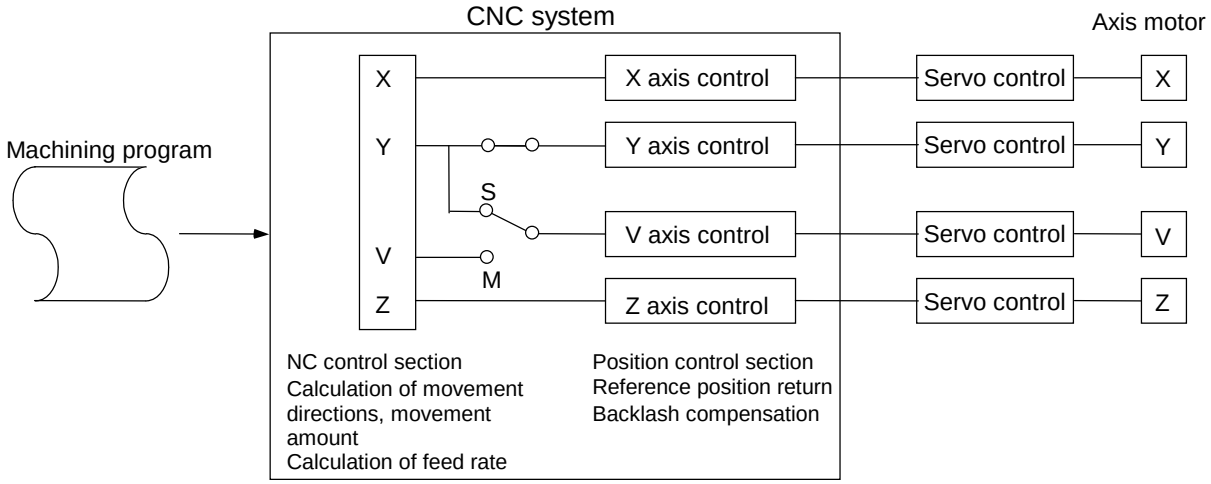
The synchronous control is a control method that both master and slave axes are controlled with the same movement command by designated the movement command for the master axis also to the slave axis. This function is assumed to be used in large machine tools, etc. which drive one axis with two servo motors.
The axis for the base of the synchronization is called the master axis, and the axis according to the master axis is called the slave axis.
The axis detach function cannot be added to the axes used in the synchronous control.

- The slave axis is controlled with the movement command for the master axis.
- One slave axis can be set to one master axis.
- Up to 3 sets of master axis / slave axis can be set in total for all the part systems.



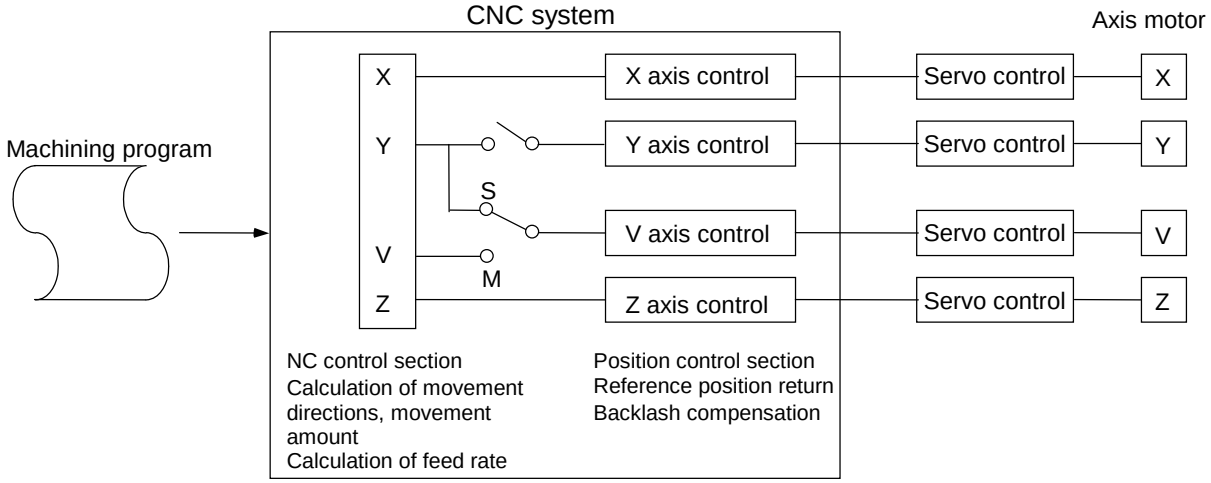
(1) Synchronous control mode
The following two operation methods are available in the synchronous control mode.

(a) Synchronous operation
This is a method that both master and slave axes are moved simultaneously with the movement command for the master axis.



There is a function that checks the correlation between the positions of the master axis and slave axis at all times while the synchronous operation method is selected to stop the feed as alarm when the error between the positions exceeds the allowable synchronization error value set in the parameter. However, when the zero point is not established, the synchronous error is not checked. Even during synchronous operation, pitch error compensation, backlash compensation and external machine coordinate compensation are performed independently for each master axis and slave axis. Designation/cancellation of synchronous operation is executed at "all axes in-position".

(b) Independent operation
This is a method that either the master or slave axis is moved with the movement command for the master axis.



Even during independent operation, pitch error compensation, backlash compensation and external machine coordinate compensation are performed independently for each master axis and slave axis. Designation/cancellation of independent operation is executed at "all axes in-position".

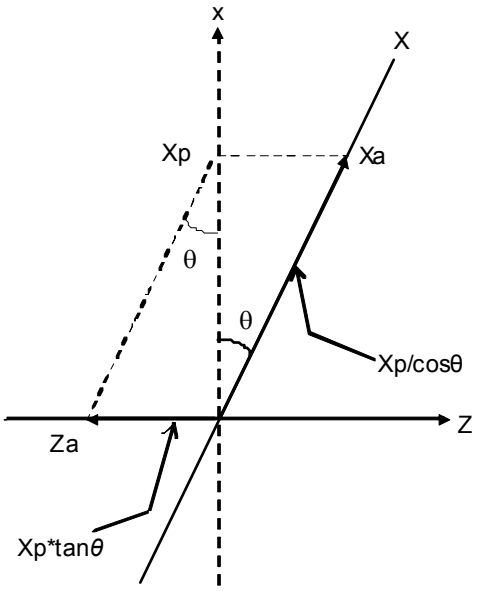
(2) Correction mode
The synchronization is temporary canceled to adjust the balance of the master and slave axes during the synchronous control mode in the machine adjustment. Each axis can be moved separately with the manual handle feed or the arbitrary feed in manual mode. If the operation mode other than the manual handle feed and arbitrary feed in manual mode is applied during the correction mode, the operation error will occur.

17.2.4 Inclined Axis Control

M system : - **L system : Δ**

Even when the control axes configuring that machine are mounted at an angle other than 90 degrees, this function enables to control by the same program as 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.
Note that the inclined axis is fixed to the 1st axis of the part system and the basic axis is fixed to the 2nd axis of the part system.

<Example of use> When the X axis serves as the inclined axis and the Z axis serves as the basic axis



X: Actual X axis
Z: Actual Z axis
x: Programmed X axis
θ: Inclination angle

Xp, the X-axis position on the programmed coordinates (on the orthogonal coordinates), is the position of Za and Xa which are produced by synthesis of Z axis and X axis.
Therefore, the X-axis (inclined axis) movement amount is expressed by the following formula:

$Xa = Xp/\cos\theta$……(1)

The Z-axis (basic axis) movement amount is compensated by the inclined movement of the X axis, and it is expressed as follows:

$Za = Zp - Xp \times \tan\theta$……..(2)

The X-axis (inclined axis) speed is as follows:

$Fa = Fp/\cos\theta$

Xa, Za and Fa are the actual movement amounts and speed. Xp, Zp and Fp are the movement amounts and speed on the program coordinates.

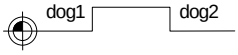
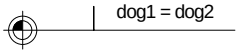
17.2.5 Position Switch

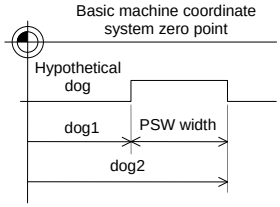
M system : ○ (16 for each part system, 16 for PLC axis) **L system : ○ (16 for each part system, 16 for PLC axis)**

Instead of a dog switch on a machine's axis, a hypothetical dog switch is established using a parameter to set a coordinate position to show the axis name and the hypothetical dog position. When the machine reaches the position, a signal is output to the PLC interface. 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 (PSW1 to PSW16). When the machine has reached the hypothetical dog positions, a signal is output to the device supported by the PLC interface.

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



17.3 PLC Operation

17.3.1 Arbitrary Feed in Manual Mode

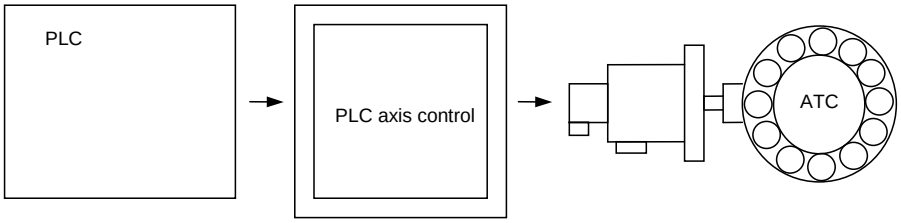
M system : ○ **L system : ○**

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 : Δ **L system : Δ**

Over and above the NC control axes, this function enables axes to be controlled independently by commands from the PLC.



Item	Details
No. of control axes	Max. 8 axes
Simultaneous control axes	The PLC control axis is controlled independently of the CNC control axis. Simultaneous start of multiple PLC axes is possible.
Command unit	Min. command unit **(Note 1)** 0.001mm (0.0001 inch) 0.0001mm (0.00001 inch)
Feedrate	0 to 1000000mm/min (0 to 100000inch/min) (The feedrate is fixed regardless of the unit system.)
Movement commands	Incremental value commands from the current position. Absolute value commands of the machine coordinate system. 0 to ±99999999 **(Note 1)**
Operation modes	Rapid traverse, cutting feed Jog feed (+), (-) Reference position return feed (+), (-) Handle feed
Backlash compensation	Provided
Stroke end	Not provided
Soft limit	Provided
Rotation axis commands	Provided Absolute value commands..... Rotation amount within one rotation. (Rotates the remainder divided by rotational axis division count.) The axis rotate in shortcut direction by the setting of a parameter "#8213 Rotation axis type". Incremental commands..... Rotates the commanded rotation amount.
Inch/mm changeover	Not provided Command to match the feedback unit.
PLC axis automatic initial set	When zero point initialization to start up the absolute position detection system uses the stopper method, the automatic initial setting operation can be selected.

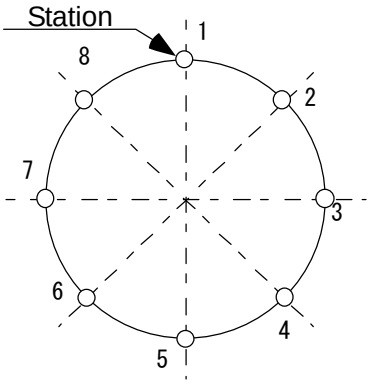
17.3.5 PLC Axis Indexing

M system : Δ **L system : Δ**

PLC axis indexing is used to move the PLC axis to the positioning destination or an arbitrary coordinate position.
This function is applied to tool exchange and magazine control.

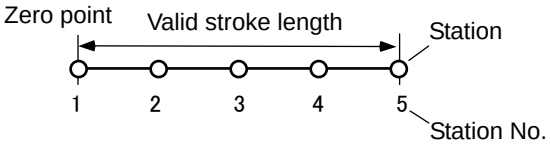
[Positioning command methods]
(1) Station method
The axis will be positioned to the destination (station) that has been decided.
There are two assigning methods: Uniform assignment and arbitrary coordinate assignment.

・Uniform assignment
One rotation (360°) of the rotary axis will be equally divided to determine the stations.
(Maximum number of divisions: 360)



[Setting 8 stations (8 divisions)]

[For linear axis]
A valid stroke will be equally divided to determine the station. (Maximum number of divisions: 359)



[Setting 5 stations]

・Arbitrary coordinate assignment
A station will be assigned to an arbitrary coordinate set in each table.

(2) Arbitrary coordinate designation method
An arbitrary coordinate will be directly designated in PLC program for positioning.

[Operation functions]

- Automatic mode
  Stations will be determined automatically.
- Manual mode
  Stations will be determined manually.
  While the start signal is ON, the axis will be rotated at a constant speed. When the start signal is OFF, the axis will be positioned at the nearest station.
- JOG mode
  The axis will be rotated at constant speed.
- Incremental feed
  The axis will be moved by the designed amount.
- Manual handle feed
  The axis will be moved by the manual pulse generator.
- Reference position return
  The axis will be positioned at the reference position.
  Reference position return is not possible by a dog switch.

[Feed functions]

- Feed rate selection
  Automatic mode and manual mode can have each four different feed rates to be designated in the PLC program.
- Acceleration/deceleration method
  Four different combination can be set from the acceleration/deceleration patterns (linear or S-pattern acceleration/deceleration) and the acceleration/deceleration time constants. The combination will be selected in the PLC program.
  Select acceleration/deceleration type with parameter: the acceleration/deceleration with constant time or the one with a constant angle of inclination.
- Short-cut control
  A least movement distance is automatically judged when a rotary axis is rotated.

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 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) 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 4 points)

For further details, refer to the PLC Interface Manual.

17.4.2 CNC Status Signal

M system : O **L system : O**

The status signals are output from the CNC system. They can be utilized by referencing them from the PLC.

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
This signal is output when the controller is reset processing.
This signal will also be output when the reset & rewind command is input to the controller, 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.

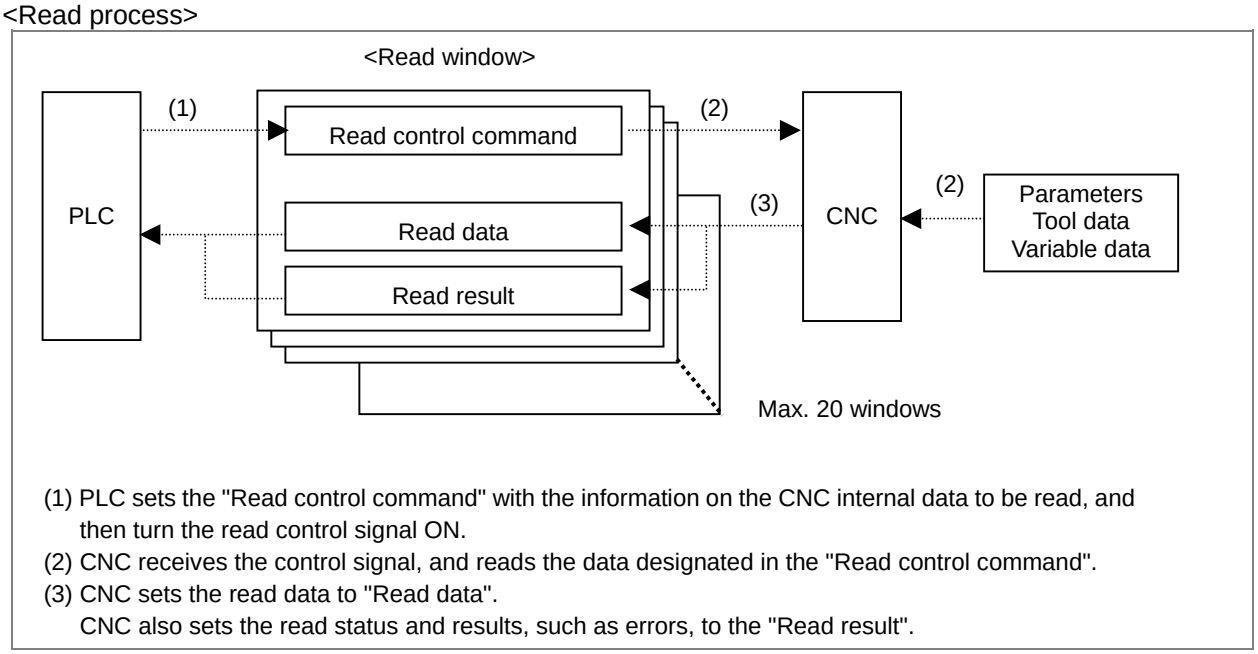
17.4.3 PLC Window

M system : Δ **L system : Δ**

PLC window is used to read/write the operation state, axis information, parameters and tool data of the CNC through a cyclic trans mission area in the CPU shared memory.

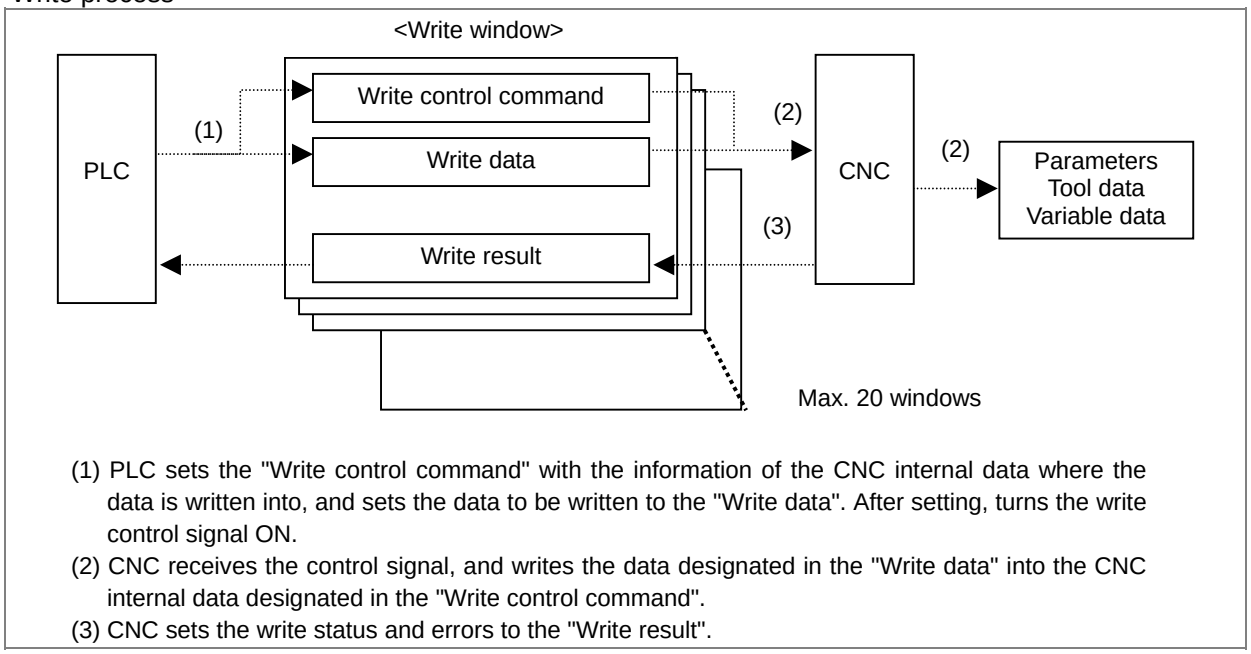
In the interface between CNC CPU for PLC window and PLC CPU, "Read control command", "Read data" and "Read result" are all called "Read window". "Write control command", "Write data" and "Write result" are all called "Write window".
These windows are used for the read and write operations. 40 units of windows, 20 units for each, are provided for "Read window" and "Write window".

Outlines of read and write processes are shown below.
<Read process>



(1) PLC sets the "Read control command" with the information on the CNC internal data to be read, and then turn the read control signal ON.
(2) CNC receives the control signal, and reads the data designated in the "Read control command".
(3) CNC sets the read data to "Read data".
CNC also sets the read status and results, such as errors, to the "Read result".

<Write process>



(1) PLC sets the "Write control command" with the information of the CNC internal data where the data is written into, and sets the data to be written to the "Write data". After setting, turns the write control signal ON.
(2) CNC receives the control signal, and writes the data designated in the "Write data" into the CNC internal data designated in the "Write control command".
(3) CNC sets the write status and errors to the "Write result".

17.4.4 External Search

M system : Δ **L system : Δ**

This function enables searching of the program to automatically start from the PLC. The program No., block No. and sequence No. can be designated. In addition, the currently searched details can be read.

17.6 External PLC Link

17.6.3 CC-Link (Master/Slave)

M system : Δ (MELSEC) L system : Δ (MELSEC)

Refer to manuals of MITSUBISHI Programmable Controller "MELSEC Q series" for information on the function and the performance.

17.6.4 PROFIBUS-DP (Master)

M system : Δ (MELSEC) L system : Δ (MELSEC)

Refer to manuals of MITSUBISHI Programmable Controller "MELSEC Q series" for information on the function and the performance.

17.6.5 DeviceNet (Master)

M system : Δ (MELSEC) L system : Δ (MELSEC)

Refer to manuals of MITSUBISHI Programmable Controller "MELSEC Q series" for information on the function and the performance.

17.6.6 FL-net

M system : Δ (MELSEC) L system : Δ (MELSEC)

Refer to manuals of MITSUBISHI Programmable Controller "MELSEC Q series" for information on the function and the performance.

17.6.7 CC-Link/LT

M system : Δ (MELSEC) L system : Δ (MELSEC)

Refer to manuals of MITSUBISHI Programmable Controller "MELSEC Q series" for information on the function and the performance.

17.6.8 CC-Link IE

M system : Δ (MELSEC) L system : Δ (MELSEC)

Refer to manuals of MITSUBISHI Programmable Controller "MELSEC Q series" for information on the function and the performance.

17.6.101 ASi

M system : Δ (MELSEC) L system : Δ (MELSEC)

Refer to manuals of MITSUBISHI Programmable Controller "MELSEC Q series" for information on the function and the performance.

17.7 Installing S/W for Machine Tools

17.7.3 EZSocket I/F (Need separate PC S/W)

M system : Δ **L system : Δ**

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 (Need separate PC S/W)

M system : Δ **L system : Δ**

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 a sequence program can be created with C language.

APLC release is activated between NC processings so that the processing frequency is not guaranteed.

[Hardware configuration]
This function will be activated by installing C language module into a built-in FROM.
The installation requires the Remote Monitor Tool.

[Software configuration]
The names of directory, file and initialize function, where C language modules are stored, are fixed.

17.8 Others

17.8.2 CNC Remote Operation Tool

17.8.2.101 Remote Monitor Tool

M system : ○ **L system : ○**

CNC remote operation tool is a PC compatible software tool that monitors information in NC unit connected with the Ethernet. (Available to download from MELFANSweb.)

17.8.102 Screen library

17.8.102.200 Cycle Monitor (Waveform Display)

M system : ○ **L system : ○**

The cycle monitor samples the PLC's device values and CNC's spindle/servo axis value, and displays the waveforms. As a sample screen of a GOT, we provide a project using GT Designer2.

Revision History

Date of revision	Manual No.	Revision details
Dec. 2006	IB(NA)1500259-A	First edition created.
Jan. 2007	IB(NA)1500259-B	Mistakes were corrected.
May 2007	IB(NA)1500259-C	The following sections are added. • 16. Drive System • 17. Machine Support Functions Other contents were added/revised/deleted according to specification.
Sep. 2010	IB(NA)1500259-D	Added/Changed the following chapters in order to support C70 software B2version. Following contents were revised. - Updated the contents of "I. GENERAL SPECIFICATIONS". - Following chapters were added to "II. FUNCTIONAL SPECIFICATIONS". 3.2.101 Hypothetical Linear Axis Control 4.5.8 High-speed Synchronous Tapping (OMR-DD) 6.1.2 Color Display(GOT) 6.2.3 Single-NC and Multi-display Unit Switch 6.2.4 Multi-NC and Common-display Unit 6.2.10 Screen Saver, Backlight OFF 6.2.15 Screen Capture 8.1.1.2 Spindle Analog I/F 11.1.2 Optional Block Skip Addition 12.1.2.102 Macro Interface Extension (1200 sets) 12.1.7.1 Chopping 12.3.1 High-speed Machining Mode Ⅰ (G5P1)[kBPM] 16.1.1.4 MDS-DM-V3/SPV2/SPV3 (200V) 17.1.2.3 Built-in PLC Processing Mode 17.1.3.6 Multi-ladder Program Register and Execution 17.1.3.7 Ladder Program Writing during RUN 17.1.3.8 PLC Protection 17.1.7.3 MELSEC Development Tool (GX Simulator) 17.1.9.1 CPU Direct Connection (RS-422/RS-232C) 17.1.9.2 CC-Link Connection (Remote device) 17.1.9.3 CC-Link Connection (Intelligent terminal) 17.5 Machine Contact Input/output I/F 17.5.1 Additional DI/DO (DI:32/DO:32) 17.6.4 PROFIBUS-DP(Master) 17.6.8 CC-Link IE 17.8.2.1 Remote monitor tool 17.8.3 Cycle Monitor (Waveform Display) - Following chapter were deleted from "II. FUNCTIONAL SPECIFICATIONS". 17.1.1 Built-in PLC Processing Mode

Date of revision	Manual No.	Revision details
Dec. 2010	IB(NA)1500259-E	Revised contents in order to support C70 software C5 version. - Added the Specifications list. - Updated the contents of "I. GENERAL SPECIFICATIONS". - Following chapters were added to "II. FUNCTIONAL SPECIFICATIONS". 1.1.4 Max. number of PLC indexing axes 1.3.102 High-speed program server mode 5.1.1.7 1000kB[2560m] (1000 programs) 5.1.1.8 2000kB[5120m] (1000 programs) 5.2.4 Word editing 8.1.7 Spindle position control (Spindle/C axis control) 17.3.5 PLC axis indexing 17.7.4 APLC release (Need separate PC S/W) - Following chapters were deleted from "II. FUNCTIONAL SPECIFICATIONS". 17.5 Machine contact input/output 17.6.102 MELSEC multiple CPU system - Following chapter Nos. were changed. 17.8.2.101 Remote monitor tool（17.8.2.1 in the former version.) 17.8.102.200 Cycle Monitor (Waveform Display)(17.8.3 in the former version.)
Jan. 2012	IB(NA)1500259-F	- Added "Handling of our product" - Mistakes were corrected.

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: +1-269-673-4092

Ohio Service Satellite
LIMA, OHIO 45801, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-847-478-2650
CLEVELAND, OHIO 44114, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-847-478-2650

Minnesota Service Satellite
MINNEAPOLIS, MINNESOTA 55413, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-847-478-2650

West Region Service Center
5665 PLAZA DRIVE, CYPRESS, CALIFORNIA 90630, U.S.A.
TEL: +1-714-220-4796 / FAX: +1-714-229-3818

East Region Service Center
200 COTTONTAIL LANE SOMERSET, NEW JERSEY 08873, U.S.A.
TEL: +1-732-560-4500 / FAX: +1-732-560-4531

Pennsylvania Service Satellite
ERIE, PENNSYLVANIA 16510, U.S.A.
TEL: +1-814-897-7820 / FAX: +1-814-987-7820

Massachusetts Service Satellite
BOSTON, MASSACHUSETTS 02108, U.S.A.
TEL: +1-508-216-6104

South Region Service Center
2810 PREMIERE PARKWAY SUITE 400, DULUTH, GEORGIA 30097, U.S.A.
TEL: +1-678-258-4500 / FAX: +1-678-258-4519

Texas Service Satellites
GRAPEVINE, TEXAS 76051, U.S.A.
TEL: +1-817-251-7468 / FAX: +1-817-416-5000
FRIENDSWOOD, TEXAS 77546, U.S.A.
TEL: +1-832-573-0787 / FAX: +1-678-573-8290

Florida Service Satellite
WEST MELBOURNE, FLORIDA 32904, U.S.A.
TEL: +1-321-610-4436 / FAX: +1-321-610-4437

Canada Region Service Center
4299 14TH AVENUE MARKHAM, ONTARIO L3R OJ2, CANADA
TEL: +1-905-475-7728 / FAX: +1-905-475-7935

Mexico City Service Center
MARIANO ESCOBEDO 69 TLALNEPANTLA, 54030 EDO. DE MEXICO
TEL: +52-55-9171-7662 / FAX: +52-55-9171-7649

Monterrey Service Satellite
MONTERREY, N.L., 64720, MEXICO
TEL: +52-81-8365-4171 / FAX: +52-81-8365-4171

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

Brazil Service Satellites
PORTO ALEGRE AND CAXIAS DO SUL BRAZIL
TEL: +55-15-3363-9927
SANTA CATARINA AND PARANA STATES
TEL: +55-15-3363-9927

EUROPE

MITSUBISHI ELECTRIC EUROPE B.V. (EUROPE FA CENTER)
GOTHAER STRASSE 10, 40880 RATINGEN, GERMANY
TEL: +49-2102-486-0 / FAX: +49-2102-486-5910

Germany Service Center
KURZE STRASSE. 40, 70794 FILDERSTADT-BONLANDEN, GERMANY
TEL: + 49-711-3270-010 / FAX: +49-711-3270-0141

France Service Center
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
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-PALAZZO SIRIO CENTRO DIREZIONALE COLLEONI,
20041 AGRATE BRIANZA MILANO ITALY
TEL: +39-039-60531-342 / FAX: +39-039-6053-206

Italy (Padova) Service Satellite
VIA SAVELLI 24 - 35129 PADOVA ITALY
TEL: +39-039-60531-342 / FAX: +39-039-6053-206

U.K. Service Center
TRAVELLERS LANE, HATFIELD, HERTFORDSHIRE, AL10 8XB, U.K.
TEL: +44-1707-27-6100 / FAX: +44-1707-27-8992

Spain Service Center
CTRA. DE RUBI, 76-80-APDO. 420
08190 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: +48-12-630-4727

Poland (Wroclaw) Service Center
UL KOBIERZYCKA 23, 52-315 WROCLAW, POLAND
TEL: +48-71-333-77-53 / FAX: +48-71-333-77-53

Turkey Service Center
BAYRAKTAR BULVARI, NUTUK SOKAK NO.5, YUKARI DUDULLU
ISTANBUL, TURKEY
TEL: +90-216-526-3990 / FAX: +90-216-526-3995

Czech Republic Service Center
TECHNOLOGICKA 374/6,708 00 OSTRAVA-PUSTKOVEC, 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

Sweden Service Center
STRANDKULLEN, 718 91 FROVI, SWEDEN
TEL: +46-581-700-20 / FAX: +46-581-700-75

Bulgaria Service Center
4 A. LYAPCHEV BOUL., 1756 - SOFIA, BULGARIA
TEL: +359-2-8176000 / 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: +380-044-494-3355 / FAX: +380-044-494-3366

Belarus Service Center
703, OKTYABRSKAYA STR., 16/5, 220030 MINSK, BELARUS
TEL: +375-17-210-4626 / FAX: +375-17-227-5830

South Africa Service Center
P.O. BOX 9234, EDLEEN, KEMPTON PARK GAUTENG, 1625 SOUTH AFRICA
TEL: +27-11-394-8512 / FAX: +27-11-394-8513

Denmark Service Center
KARETMAGERVEJ. 7A, DK-7000, FREDERICIA, DENMARK
TEL: +45-7620-7514

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

Indonesia Service Center
THE PLAZZA OFFICE TOWER, 28TH FLOOR JL.M.H. THAMRIN KAV.28-30, JAKARTA, INDONESIA
TEL: +62-21-2992-2333 / FAX: +62-21-2992-2555

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
NO. 16, JALAN SHAH BANDAR 1, TAMAN UNGKU TUN AMINAH, 81300 SKUDAI, JOHOR MALAYSIA
TEL: +60-7-557-8218 / FAX: +60-7-557-3404

Vietnam Service Center-1
ROOM 1004, 1005, FLOOR 10, 255 TRAN HUNG DAO CO GIANG WARD, DIST. 1, HCMC, VIETNAM
TEL: +84-8-3838-6931 / FAX: +84-8-3838-6932

Vietnam Service Center-2
LOT G10 - AREA 4 - HIEP BINH CHANH WARD - THU DUC DISTRICT - HCMC, VIETNAM
TEL: +84-8-2240-3587 / FAX: +84-8-3726-7968

Vietnam (Hanoi) Service Center
5FL, 59 - XA DAN STR., DONG DA DIST., HN, VIETNAM
TEL: +84-4-3573-7646 / FAX: +84-4-3573-7650

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

MITSUBISHI ELECTRIC AUTOMATION (THAILAND) CO., LTD. (THAILAND FA CENTER)

BANG-CHAN INDUSTRIAL ESTATE NO.111 SOI SERITHAI 54
T.KANNAYAO, A.KANNAYAO, BANGKOK 10230, THAILAND
TEL: +66-2906-8255 / FAX: +66-2906-3239

Thailand Service Center
898/19,20,21,22 S.V. CITY BUILDING OFFICE TOWER 1, FLOOR 7
RAMA III RD., BANGPONGPANG, YANNAWA, BANGKOK 10120, THAILAND
TEL: +66-2-682-6522 / FAX: +66-2-682-9750

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: +91-124-4630300 / FAX: +91-124-4630399

India (Bangalore) Service Center
FIRST & SECOND FLOOR, AVR BASE, MUNICIPAL NO.BC-308,
HENNURE BANASWADI ROAD, HRBR RING ROAD, BANGALORE-560 043, INDIA
TEL: +91-80-4020-1600 / FAX: +91-80-4020-1699
Chennai satellite office
Coimbatore satellite office

India (Pune) Service Center
TEL: +91-998-7997651
Baroda satellite office

OCEANIA

MITSUBISHI ELECTRIC AUSTRALIA LTD.

Oceania 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: +86-21-2308-2830
China (Ningbo) Service Dealer
China (Wuxi) Service Dealer
China (Jinan) 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: +86-10-6518-3907
China (Beijing) Service Dealer

China (Tianjin) Service Center
B-2 801/802, YOUYI BUILDING, NO.50 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
1480-6, GAYANG-DONG, GANGSEO-GU SEOUL 157-200, KOREA
TEL: +82-2-3660-9602 / FAX: +82-2-3664-8668

Korea Taegu Service Satellite
603 CRYSTAL BUILDING 1666, SANBYEOK-DONG, BUK-KU, DAEGU, 702-010, KOREA
TEL: +82-53-604-6047 / FAX: +82-53-604-6049

TAIWAN

MITSUBISHI ELECTRIC TAIWAN CO., LTD. (TAIWAN FA CENTER)

Taiwan (Taichung) Service Center
NO.8-1, GONG YEH 16TH RD., TAICHUNG INDUSTRIAL PARK TAICHUNG CITY, TAIWAN R.O.C.
TEL: +886-4-2359-0688 / FAX: +886-4-2359-0689

Taiwan (Taipei) Service Center
3RD. FLOOR, NO.122 WUKUNG 2ND RD., WU-KU HSIANG, TAIPEI HSIEN, TAIWAN R.O.C.
TEL: +886-2-2299-2205 / FAX: +886-2-2298-1909

Taiwan (Tainan) Service Center
2F(C),1-1, CHUNGHWA-RD., YONGKANG CITY, TAINAN HSIEN, TAIWAN R.O.C.
TEL: +886-6-313-9600 / FAX: +886-6-313-7713

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

MITSUBISHI CNC

MITSUBISHI ELECTRIC CORPORATION
HEAD OFFICE : TOKYO BUILDING,2-7-3 MARUNOUCHI,CHIYODA-KU,TOKYO 100-8310,JAPAN

MODEL	C70
MODEL CODE	100-009
Manual No.	IB-1500259

IB(NA)1500259(ENG)-F(1201)MEE
Made in JAPAN