

CNC

MELDAS 60/60S Series

SPECIFICATIONS MANUAL



MELDAS and MELSEC are registered trademarks of Mitsubishi Electric Corporation.

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

Introduction




This manual describes the specifications of MELDAS 60/60S Series.
To safely use this CNC unit, thoroughly study the "Precautions for Safety" on the next page before use.

Details described in this manual

At the beginning of each item, a table indicating its specification according to the model.

- : Standard
- △ : Option
- : Selection
- ☆ : Special option

CAUTION

-  The items that are not described in this manual must be interpreted as "not possible".
-  This manual is written on the assumption that all option functions are added.
-  Some functions may differ or some functions may not be usable depending on the NC system (software) version.

General precautions

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

Precautions for Safety

Always read 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".




When there is a great risk that the user could be subject to fatalities or serious injuries if handling is mistaken.



When the user could be subject to fatalities or serious injuries if handling is mistaken.



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.
-  Follow the remote type machine contact input/output interface described in this manual. (Connect a diode in parallel with the inductive load or connect a protective resistor in serial with the capacitive load, etc.)
-  If the parameter is used to set the temperature rise detection function to invalid, overheating may occur, thereby disabling control and possibly resulting in the axes running out of control, which in turn may result in machine damage and/or bodily injury or destruction of the unit. It is for this reason that the detection function is normally left "valid" for operation.

CONTENTS

1 Control Axes	1
1.1 Control Axes	1
1.1.1 Number of Basic Control Axes (NC axes)	1
1.1.2 Max. Number of Control Axes (NC axes + Spindles + PLC axes + Auxiliary axes).....	1
1.1.3 Number of Simultaneous Contouring Control Axes	3
1.1.4 Max. Number of NC Axes in a Part System (Part System 1/Part System 2)	3
1.2 Control Part System.....	3
1.2.1 Standard Number of Part Systems.....	3
1.2.2 Max. Number of Part Systems.....	3
1.3 Control Axes and Operation Modes	4
1.3.1 Tape (RS-232C Input) Mode	4
1.3.2 Memory Mode	4
1.3.3 MDI Mode.....	4
1.3.4 High-Speed Program Server Mode	4
1.3.5 IC Card Mode.....	4
1.3.5.1 IC Card in Control Unit.....	4
2 Input Command	5
2.1 Data Increment	5
2.2 Unit System.....	6
2.2.1 Inch/Metric Changeover.....	6
2.3 Program Format.....	7
2.3.1 Character Code.....	7
2.3.2 Program Format.....	8
2.3.2.1 Format 1 for Lathe	8
2.3.2.3 Special Format for Lathe	8
2.3.2.4 Format 1 for Machining Center.....	8
2.3.2.7 Special Format for MELDAS	8
2.4 Command Value	9
2.4.1 Decimal Point Input I, II.....	9
2.4.2 Absolute / Incremental Command	10
2.4.3 Diameter/Radius Designation.....	12
2.5 Command Value and Setting Value Range	13
2.5.1 Command Value and Setting Value Range	13
3 Positioning / Interpolation	17
3.1 Positioning	17
3.1.1 Positioning.....	17
3.1.2 Unidirectional Positioning	18
3.2 Linear/Circular Interpolation	19
3.2.1 Linear Interpolation	19
3.2.2 Circular Interpolation (Center / Radius Designation)	20
3.2.3 Helical Interpolation	22
3.2.4 Spiral / Conical Interpolation.....	24
3.2.5 Cylindrical Interpolation.....	26
3.2.6 Polar coordinate interpolation	27
3.2.7 Milling Interpolation	28
3.3 Curve Interpolation	29
3.3.2 Exponential Interpolation	29
3.3.3 Spline Interpolation	30

4 Feed	31
4.1 Feed Rate	31
4.1.1 Rapid Traverse Rate (m/min)	31
4.1.2 Cutting Feed Rate (m/min)	32
4.1.3 Manual Feed Rate (m/min).....	33
4.2 Feed Rate Input Methods.....	34
4.2.1 Feed per Minute.....	34
4.2.2 Feed per Revolution.....	36
4.2.4 F 1-digit Feed.....	37
4.3 Override	38
4.3.1 Rapid Traverse Override	38
4.3.2 Cutting Feed Override	38
4.3.3 2nd Cutting Feed Override	39
4.3.4 Override Cancel.....	39
4.4 Acceleration / Deceleration.....	40
4.4.1 Automatic Acceleration / Deceleration after Interpolation.....	40
4.4.2 Rapid Traverse Constant Inclination Acceleration / Deceleration	41
4.4.3 Rapid Traverse Constant Inclination 3-step Acceleration / Deceleration	44
4.5 Thread Cutting	45
4.5.1 Thread Cutting (Lead/Thread Number Designation).....	45
4.5.2 Variable Lead Thread Cutting.....	47
4.5.3 Synchronous Tapping.....	48
4.5.3.1 Synchronous Tapping Cycle.....	48
4.5.3.2 Pecking Tapping Cycle.....	50
4.5.3.3 Deep-hole Tapping Cycle.....	51
4.5.4 Chamfering.....	52
4.5.6 Circular Thread Cutting.....	53
4.6 Manual Feed.....	54
4.6.1 Manual Rapid Traverse	54
4.6.2 Jog Feed	54
4.6.3 Incremental Feed.....	55
4.6.4 Handle Feed.....	55
4.6.5 Manual Feed Rate B.....	56
4.7 Dwell	57
4.7.1 Dwell (Time-based Designation)	57
5 Program Memory / Editing	58
5.1 Memory Capacity.....	58
5.1.1 Memory Capacity (Number of Programs Stored).....	58
5.2 Editing Method.....	59
5.2.1 Program Editing	59
5.2.2 Background Editing.....	60
5.2.3 Buffer Correction.....	61
5.2.4 Word Editing.....	62
6 Operation and Display	63
6.1 Structure of Operation / Display Panel.....	63
6.2 Operation Methods and Functions.....	63
6.2.1 Memory Switch (PLC Switch).....	63
6.3 Display Methods and Contents	64
6.3.1 Status Display	64
6.3.2 Position Display	64
6.3.3 Program Running State Display	66
6.3.4 Parameter Setting and Display.....	66
6.3.5 MDI Data Setting and Display	66
6.3.6 Specification List Display	66

6.3.7 Clock	66
6.3.8 Hardware / Software Configuration Display	67
6.3.9 Integrated Time Display	67
6.3.10 Available Languages	67
6.3.11 Additional Languages	68
6.3.11.1 Japanese	68
6.3.11.2 English	68
6.3.11.3 German	68
6.3.11.4 Italian.....	68
6.3.11.5 French	68
6.3.11.6 Spanish	68
6.3.11.7 Chinese	69
6.3.11.8 Korean.....	69
6.3.11.9 Portuguese.....	69
6.3.11.10 Hungarian	69
6.3.11.11 Dutch.....	69
6.3.11.12 Swedish	69
6.3.12 Screen Saver, Backlight OFF	70
6.3.13 Screen Deletion	70
7 Input / Output Functions and Devices	71
7.1 Input / Output Data	71
7.2 Input / Output I/F	72
7.2.1 RS-232C I/F	72
7.2.2 IC Card I/F.....	72
7.2.2.1 I/F for IC Card in Control Unit	72
7.2.3 Data Server I/F.....	72
7.2.4 Ethernet I/F	72
7.3 Computer Link.....	73
7.3.1 Computer Link B	73
8 Spindle, Tool and Miscellaneous Functions	74
8.1 Spindle Functions (S)	74
8.1.1 Command / Output	74
8.1.1.1 Spindle Functions	74
8.1.1.2 Spindle Serial I/F.....	75
8.1.1.3 Spindle Analog I/F.....	75
8.1.1.4 Coil Change	75
8.1.1.5 Automatic Coil Change	75
8.1.2 Speed Control	76
8.1.2.1 Constant Surface Speed Control.....	76
8.1.2.2 Spindle Override	77
8.1.2.3 Multiple-spindle Control	77
8.1.2.3.1 Multiple-spindle Control I	78
8.1.2.3.2 Multiple-spindle Control II.....	78
8.1.3 Position Control.....	79
8.1.3.1 Spindle Orientation	79
8.1.3.2 Spindle Position Control (Spindle / C Axis Control)	79
8.1.3.3 Spindle Synchronization	80
8.1.3.3.1 Spindle Synchronization I.....	80
8.1.3.3.2 Spindle Synchronization II.....	81
8.1.3.4 Tool Spindle Synchronization I (Polygon)	81
8.1.3.4.1 Tool Spindle Synchronization IA (Spindle-Spindle, Polygon).....	81
8.1.3.4.2 Tool Spindle Synchronization IB (Spindle-Spindle, Polygon).....	82
8.1.3.4.3 Tool Spindle Synchronization IC (Spindle-NC axis, Polygon).....	83

8.2 Tool Functions (T).....	84
8.2.1 Tool Functions.....	84
8.3 Miscellaneous Functions (M).....	85
8.3.1 Miscellaneous Functions	85
8.3.2 Multiple M Codes in 1 Block	85
8.3.3 M Code Independent Output	86
8.3.4 Miscellaneous Function Finish	86
8.3.5 M Code Output during Axis Positioning.....	87
8.4 2nd Miscellaneous Functions (B)	88
8.4.1 2nd Miscellaneous Functions	88
9 Tool Compensation	89
9.1 Tool Length / Position Offset	89
9.1.1 Tool Length Offset	89
9.1.2 Tool Position Offset.....	92
9.1.3 Tool Offset for Additional Axes	92
9.2 Tool Radius	93
9.2.1 Tool radius Compensation.....	93
9.2.2 3-dimensional Tool Radius Compensation	96
9.2.3 Tool Nose Radius Compensation (G40/41/42)	97
9.2.4 Automatic Decision of Nose Radius Compensation Direction (G46/40)	98
9.3 Tool Offset Amount.....	99
9.3.1 Number of Tool Offset Sets	99
9.3.1.1 20	99
9.3.1.2 40	99
9.3.1.3 80	99
9.3.1.5 200	99
9.3.1.6 400	99
9.3.1.7 999	99
9.3.2 Offset Memory	100
9.3.2.1 Tool Shape/Wear Offset Amount	100
10 Coordinate System.....	103
10.1 Coordinate System Type and Setting	103
10.1.1 Machine Coordinate System	104
10.1.2 Coordinate System Setting.....	105
10.1.3 Automatic Coordinate System Setting.....	106
10.1.4 Workpiece Coordinate System Selection (6 sets).....	107
10.1.5 Extended Workpiece Coordinates System Selection	108
10.1.6 Workpiece Coordinate System Preset (G92.1)	109
10.1.7 Local Coordinate System	110
10.1.8 Coordinate System for Rotary Axis	111
10.1.9 Plane Selection	111
10.1.10 Origin Set	112
10.1.11 Counter Set.....	112
10.2 Return	113
10.2.1 Manual Reference Position Return	113
10.2.2 Automatic 1st Reference Position Return	114
10.2.3 2nd, 3rd, 4th Reference Position Return	116
10.2.4 Reference Position Verification	117
10.2.5 Absolute position detection.....	118
10.2.6 Tool Exchange Position Return	119
10.2.7 C Axis Reference Position Return.....	120

11 Operation Support Functions	122
11.1 Program Control	122
11.1.1 Optional Block Skip.....	122
11.1.3 Single Block	123
11.2 Program Test	124
11.2.1 Dry Run.....	124
11.2.2 Machine Lock.....	124
11.2.3 Miscellaneous Function Lock	125
11.2.4 Graphic Check	125
11.2.5 Graphic Trace	125
11.3 Program Search / Start / Stop	126
11.3.1 Program Search.....	126
11.3.2 Sequence Number Search	126
11.3.3 Collation Stop.....	127
11.3.4 Program Restart.....	128
11.3.5 Automatic Operation Start	129
11.3.6 NC Reset.....	129
11.3.7 Feed Hold.....	130
11.3.8 Search & Start.....	130
11.4 Interrupt Operation.....	131
11.4.1 Manual Interruption.....	131
11.4.2 Automatic Operation Handle Interruption	132
11.4.3 Manual Absolute Mode ON / OFF	133
11.4.4 Thread Cutting Cycle Retract	134
11.4.5 Tapping Retract	135
11.4.6 Manual Numerical value Command	136
11.4.8 MDI Interruption	136
11.4.9 Simultaneous Operation of Manual and Automatic Modes	137
11.4.10 Simultaneous Operation of Jog and Handle Modes	137
11.4.11 Reference Position Retract.....	138
11.4.13 Tool Escape and Return.....	138
11.4.14 Skip Retract.....	138
12 Program Support Functions.....	139
12.1 Machining Method Support Functions.....	139
12.1.1 Program	139
12.1.1.1 Subprogram Control	139
12.1.1.3 Scaling	141
12.1.2 Macro Program	142
12.1.2.1 User Macro	142
12.1.2.2 Machine Tool Builder Macro.....	144
12.1.2.2.1 Machine Tool Builder Macro SRAM.....	144
12.1.2.3 Macro Interruption.....	145
12.1.2.4 Variable Command.....	146
12.1.2.4.1 100 sets	146
12.1.2.4.2 200 sets	146
12.1.2.4.3 300 sets	147
12.1.2.4.4 600 sets	147
12.1.2.4.5 700 sets	147
12.1.2.4.6 (50+50 x number of part systems) sets	147
12.1.2.4.7 (100+100 x number of part systems) sets	147
12.1.2.4.8 (200+100 x number of part systems) sets	147
12.1.2.4.9 (400+100 x number of part systems) sets	147
12.1.2.4.10 (500+100 x number of part systems) sets	147

12.1.3 Fixed Cycle	148
12.1.3.1 Fixed Cycle for Drilling	149
12.1.3.2 Special Fixed Cycle	155
12.1.3.3 Fixed Cycle for Turning Machining	159
12.1.3.4 Multiple Repetitive Fixed Cycle for Turning Machining	164
12.1.3.5 Multiple Repetitive Fixed Cycle for Turning Machining (Type II)	172
12.1.3.6 Small-diameter Deep-Hole Drilling Cycle	173
12.1.3.7 Fixed Cycle for Drilling (Type II)	174
12.1.4 Mirror Image	175
12.1.4.1 Mirror Image by Parameter Setting	175
12.1.4.2 External Input Mirror Image	175
12.1.4.3 G Code Mirror Image	176
12.1.4.4 Mirror Image for Facing Tool Posts	177
12.1.4.5 T Code Mirror Image for Facing Tool Posts	177
12.1.5 Coordinate System Operation	178
12.1.5.1 Coordinate Rotation by Program	178
12.1.6 Dimension Input	180
12.1.6.1 Corner Chamfering / Corner R	180
12.1.6.2 Linear Angle Command	186
12.1.6.3 Geometric Command	187
12.1.6.4 Polar Coordinate Command	191
12.1.7 Axis Control	192
12.1.7.1 High-speed Machining	192
12.1.7.1.3 High-speed Machining Mode III	192
12.1.7.1.4 High-speed High-accuracy Control 1	193
12.1.7.1.5 High-speed High-accuracy Control 2	194
12.1.7.1.7 SSS Control	196
12.1.7.2 Chopping	197
12.1.7.2.1 Chopping	197
12.1.7.4 Normal Line Control	199
12.1.7.5 Circular Cutting	200
12.1.8 Multi-part System Control	201
12.1.8.1 Synchronization between Part Systems	201
12.1.8.2 Start Point Designation Synchronization	202
12.1.8.3 Cross Machining Command	204
12.1.8.5 Control Axis Synchronization	205
12.1.8.6 Balance Cut	206
12.1.8.7 Common Memory between Tool Posts	207
12.1.8.9 2-part System Program Management	207
12.1.9 Data Input by Program	208
12.1.9.1 Parameter Input by Program	208
12.1.9.2 Compensation Data Input by Program	209
12.1.10 Machining Modal	211
12.1.10.1 Tapping Mode	211
12.1.10.2 Cutting Mode	211
12.2 Machining Accuracy Support Functions	212
12.2.1 Automatic Corner Override	212
12.2.2 Deceleration Check	213
12.2.2.1 Exact Stop Mode	214
12.2.2.2 Exact Stop Check	214
12.2.2.3 Error Detect	214
12.2.2.4 Programmable Inposition Check	215
12.2.3 High-Accuracy Control (G61.1)	216
12.3 Programming Support Functions	219
12.3.1 Playback	219
12.3.2 Address Check	219

13 Machine Accuracy Compensation	220
13.1 Static Accuracy Compensation	220
13.1.1 Backlash Compensation	220
13.1.2 Memory-type Pitch Error Compensation	221
13.1.3 Memory-type Relative Position Error Compensation	222
13.1.4 External Machine Coordinate System Compensation	223
13.1.5 Circular Radius Error Compensation	223
13.2 Dynamic Accuracy Compensation	224
13.2.1 Smooth High-gain Control (SHG Control)	224
13.2.2 Dual Feedback	225
13.2.3 Lost Motion Compensation	225
14 Automation Support Functions	226
14.1 External Data Input	226
14.1.1 External Search	226
14.1.2 External Workpiece Coordinate Offset	227
14.1.3 External Tool Offset	228
14.2 Measurement	229
14.2.1 Skip	229
14.2.1.1 Skip	229
14.2.1.2 Multiple-step Skip	230
14.2.1.4 PLC Skip	231
14.2.1.5 Speed Change Skip	232
14.2.5 Automatic Tool Length Measurement	233
14.2.6 Manual Tool Length Measurement 1	236
14.2.7 Manual Tool Length Measurement 2	237
14.2.8 Workpiece Coordinate Offset Measurement	238
14.2.9 Workpiece Position Measurement	239
14.3 Monitoring	241
14.3.1 Tool Life Management	241
14.3.2 Number of Tool Life Management Sets	242
14.3.3 Display of Integrated Time/Number of Parts	242
14.3.4 Load Meter	242
14.3.5 Position Switch	243
14.5 Others	244
14.5.1 Programmable Current Limitation	244
15 Safety and Maintenance	245
15.1 Safety Switches	245
15.1.1 Emergency Stop	245
15.1.2 Data Protection Key	245
15.2 Display for Ensuring Safety	246
15.2.1 NC Warning	246
15.2.2 NC Alarm	246
15.2.3 Operation Stop Cause	247
15.2.4 Emergency Stop Cause	247
15.2.5 Temperature Detection	248
15.3 Protection	249
15.3.1 Stroke End (Over Travel)	249
15.3.2 Stored Stroke Limit	249
15.3.2.1 Stored Stroke Limit I/II	250
15.3.2.2 Stored Stroke Limit IB	252
15.3.2.3 Stored Stroke Limit IIB	253
15.3.2.4 Stored Stroke Limit IC	253
15.3.4 Chuck/Tail Stock Barrier Check	254
15.3.5 Interlock	255
15.3.6 External Deceleration	255

15.3.8 Door Interlock.....	256
15.3.8.1 Door Interlock I.....	256
15.3.8.2 Door Interlock II.....	257
15.3.9 Parameter Lock.....	258
15.3.10 Program Protect (Edit Lock B, C)	258
15.3.11 Program Display Lock.....	259
15.4 Maintenance and Troubleshooting.....	260
15.4.1 History Diagnosis	260
15.4.2 Setup / Monitor for Servo and Spindle	260
15.4.3 Data Sampling	260
15.4.4 Waveform Display.....	261
15.4.5 Machine Operation History Monitor	261
15.4.6 NC Data Backup	262
15.4.7 PLC I/F Diagnosis	263
15.4.8 MELDASNET	264
15.4.8.1 Machine Builder Network System	264
15.4.8.2 Anshin-net.....	264
16 Cabinet and Installation.....	265
16.1 Cabinet Construction	265
16.2 Power Supply, Environment and Installation Conditions	269
17 Servo/Spindle System.....	272
17.1 Feed Axis	272
17.1.1 MDS-C1-V1/C1-V2 (200V)	272
17.1.2 MDS-B-V14L (200V).....	272
17.1.3 MDS-CH-V1/CH-V2 (400V).....	272
17.1.4 MDS-B-SVJ2 (Compact and Small Capacity).....	273
17.2 Spindle	274
17.2.1 MDS-C1-SP/C1-SPM/B-SP (200V).....	274
17.2.2 MDS-CH-SP/CH-SPH (400V)	274
17.2.3 MDS-B-SPJ2 (Compact and Small Capacity).....	274
17.3 Auxiliary Axis.....	274
17.3.1 Index/Positioning Servo: MR-J2-CT	274
17.4 Power Supply.....	275
17.4.1 Power Supply: MDS-C1-CV/B-CVE	275
17.4.2 AC Reactor for Power Supply.....	275
17.4.3 Ground plate	275
17.4.4 Power Supply: MDS-A-CR (Resistance Regeneration).....	275
18 Machine Support Functions.....	276
18.1 PLC	276
18.1.1 PLC Basic Function	276
18.1.1.1 Built-in PLC Basic Function	276
18.1.2 Built-in PLC Processing Mode.....	281
18.1.2.1 PLC4B I/F	281
18.1.2.2 MELSEC Development Tool I/F	281
18.1.3 Built-in PLC Capacity (Number of Steps).....	282
18.1.4 Machine Contact Input/Output I/F.....	282
18.1.5 Ladder Monitor.....	287
18.1.6 PLC Development.....	287
18.1.6.1 On-board Development	287
18.1.6.2 MELSEC Development Tool	287
18.1.6.3 PLC4B.....	287
18.1.7 C Language Function	287
18.1.8 PLC Protect.....	288
18.1.9 PLC Password Lock	289

18.1.12 GOT Connection	290
18.1.12.2 CC-Link Connection (Remote Device)	290
18.1.13 PLC Message	291
18.1.13.1 Japanese	291
18.1.13.2 English	291
18.1.13.3 German	291
18.1.13.4 Italian.....	291
18.1.13.5 French.....	291
18.1.13.6 Spanish	291
18.1.13.9 Portuguese.....	291
18.1.13.10 Hungarian	292
18.1.13.11 Dutch.....	292
18.1.13.12 Swedish	292
18.2 Machine Construction.....	293
18.2.1 Servo OFF.....	293
18.2.2 Axis Detach.....	294
18.2.3 Synchronous Control	295
18.2.3.1 Position Tandem.....	295
18.2.4 Inclined Axis Control	297
18.2.5 Index Table Indexing	298
18.2.6 NSK Table Connection Control	299
18.2.7 Auxiliary Axis Control.....	300
18.3 PLC Operation	301
18.3.1 Arbitrary Feed in Manual Mode	301
18.3.3 PLC Axis Control.....	302
18.4 PLC Interface	303
18.4.1 CNC Control Signal	303
18.4.2 CNC Status Signal.....	304
18.4.5 DDB.....	306
18.5 Machine Contact I / O.....	307
18.6 External PLC Link.....	308
18.6.1 External PLC link I (M-NET)	308
18.6.2 External PLC Link II (Bus Connection).....	310
18.6.2.1 MELSEC-A, QnA Series.....	310
18.6.3 IO Link.....	311
18.6.4 CC-Link	312
18.7 Installing S/W for Machine Tools.....	316
18.7.1 APLC.....	316

Appendix 1 List of Specifications..... 317

Appendix 2 Outline and installation dimension drawings of units 318

Appendix 2.1 Outline drawing of control unit.....	318
Appendix 2.1 Outline Drawing of Control Unit.....	318
Appendix 2.1.1 M64A/M64 Control Unit Outline Drawing	318
Appendix 2.1.2 M64AS/64S/65/65S/66/66S Control Unit Outline Drawing.....	319
Appendix 2.2 Communication Terminal Outline Drawing	320
Appendix 2.2.1 FCUA-CT100/CT120.....	320
Appendix 2.2.2 FCUA-CR10.....	321
Appendix 2.2.3 FCUA-LD100	322
Appendix 2.2.4 FCUA-LD10	323
Appendix 2.2.5 FCU6-DUT32.....	324
Appendix 2.2.6 FCU6-DUT33.....	325
Appendix 2.2.7 FCUA-KB20/KB30.....	326
Appendix 2.2.8 FCU6-KB021/KB031	327
Appendix 2.2.9 FCUA-KB10/KB12.....	328
Appendix 2.2.10 Key Arrangement	329
Appendix 2.3 Base I/O Unit Outline Drawing	330
Appendix 2.4 Remote I/O Unit Outline Drawing.....	331

1 Control Axes

1.1 Control Axes

1 Control Axes

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

The NC axis is an axis that can be manually operated, or automatically operated with the machining program.

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

1.1 Control Axes

1.1.1 Number of Basic Control Axes (NC axes)

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○3	○3	○3	○3	○3	○3	○3	○3
L system	○2	○2	○2	○2	○2	○2	○2	○2

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

Max. number of control axes (NC axes + spindles + PLC axes + auxiliary axes)

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	9	11	11	11	9	11	11	11
L system	9	11	11	11	9	11	14	14

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, PLC axis and auxiliary axis can be used.

Connection specifications of NC axis, PLC axis, spindle and auxiliary axis

There are two channels with which the servo and spindle are connected.

Maximum 7 axes can be connected with each channel.

NC axis, PLC axis, spindle : They can be connected with the channel 1 and channel 2. The connection number of first axis to seventh axis are assigned to each channel. Connect them from the first axis in order. More than one axis must be connected with the channel 1.

Auxiliary axis (J2-CT) : They can be connected with the channel 2. The connection order is the NC axis, PLC axis, spindle and auxiliary axis.

1 Control Axes

1.1 Control Axes

Max. number of axes (NC axes + spindles + PLC axes)

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	5	7	7	7	5	7	7	7
L system	5	7	7	7	5	7	14	14

Max number of servo axes (NC axes + PLC axes)

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	5	6	6	6	5	7	7	7
L system	5	6	6	6	5	7	14	14

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

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	4	6	6	6	4	6	6	6
L system	4	6	6	6	4	7	12	12

Max. number of spindles

Includes analog spindles.

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	1	2	2	2	1	2	4	4
L system	1	2	2	2	1	2	4	4

Max. number of PLC axes

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	2	2	2	2	2	2	2	2
L system	2	2	2	2	2	2	2	2

Max. number of auxiliary axes

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	4	4	4	4	4	4	4	4
L system	4	4	4	4	4	4	4	4

1 Control Axes

1.1 Control Axes

1.1.3 Number of Simultaneous Contouring Control Axes

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

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	4	4	4	6	4	4	4	6
L system	4	4	4	6	4	4	4	6

1.1.4 Max. Number of NC Axes in a Part System

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	4	6	6	6	4	6	6	6
L system	4	6	6	6	4	6	6	6

1.2 Control Part System

1.2.1 Standard Number of Part Systems

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	1	1	1	1	1	1	1	1
L system	1	1	1	1	1	1	1	1

1.2.2 Max. Number of Part Systems

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	1Δ	2Δ	2Δ	2Δ	1Δ	1Δ	1Δ	1Δ
L system	1Δ	2Δ	2Δ	2Δ	1Δ	2Δ	2Δ	2Δ

For actual use, the machine maker specification will apply.

1 Control Axes

1.3 Control Axes and Operation Modes

1.3 Control Axes and Operation Modes

1.3.1 Tape (RS-232C Input) Mode

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		○	○	○	○	○	○	○	○
M system		○	○	○	○	○	○	○	○
L system		○	○	○	○	○	○	○	○

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

1.3.2 Memory Mode

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		○	○	○	○	○	○	○	○
M system		○	○	○	○	○	○	○	○
L system		○	○	○	○	○	○	○	○

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

1.3.3 MDI Mode

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		○	○	○	○	○	○	○	○
M system		○	○	○	○	○	○	○	○
L system		○	○	○	○	○	○	○	○

The MDI data stored in the memory of the NC 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.

1.3.4 High-Speed Program Server Mode

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		–	–	Δ	Δ	–	Δ	Δ	Δ
M system		–	–	Δ	Δ	–	Δ	Δ	Δ
L system		–	–	–	–	–	–	–	–

In this mode, the NC unit is hooked up to a host computer using an Ethernet connection, and the programs are transferred to either the NC unit's memory or the IC card inside the NC unit. This function is executed with the transferred programs and memory operation (IC card operation) mode.

1.3.5 IC Card Mode

1.3.5.1 IC Card in Control Unit

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		–	–	Δ	Δ	–	Δ	Δ	Δ
M system		–	–	Δ	Δ	–	Δ	Δ	Δ
L system		–	–	–	–	–	–	–	–

The machine programs stored in the IC card of the NC unit are run.

2 Input Command

2.1 Data Increment

2 Input Command

2.1 Data Increment

Least command increment: 1 μm

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

Least command increment: 0.1 μm

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	△	△	△	△	△	△	△	△
L system	△	△	△	△	△	△	△	△

Least command increment: 0.01 μm (10nm)

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	–	–	–	–	–	–	–	–
L system	–	–	–	–	–	–	△	△

(Note 1) The drive unit must be compatible to use the least command increment 0.01 μm (10nm). The relation of the control unit setting and usable drive unit compliance is shown below.

Last command increment	MDS-B series	MDS-C1 series
1 μm /0.1 μm	○	○
0.01 μm (10nm)	–	○ (May be incompatible depending on servo drive unit version)

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 (1st part system, 2nd part system).
- (2) The least command increment indicates the command increment of the movement command in the machining program. This can be set per axis.

Increment type	Type	Metric unit system		Inch unit system	
		Linear axis (Unit = mm)	Rotary axis (Unit = °)	Linear axis (Unit = inch)	Rotary axis (Unit = °)
Least command increment	B	0.001	0.001	0.0001	0.001
	C	0.0001	0.0001	0.00001	0.0001
	D	0.00001	0.00001	–	–

(Note) 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.
- (4) If the least command increment is 0.1 μm /0.01 μm , the movement amount, movement range and command speed may be limited compared to the 1 μm .

2 Input Command

2.2 Unit System

2.2 Unit System

2.2.1 Inch/Metric Changeover

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ

The unit systems of the data handled in the controller include the metric unit system and inch unit 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) For the angle data, 1.0 means 1 degree (°) regardless of the unit system.

Data \ Parameter		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 Input Command

2.3 Program Format

2.3 Program Format

2.3.1 Character Code

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

The command information used in this CNC system consists of alphanumerics and symbols which are collectively known as characters.

These characters are expressed as combinations of 8-bit data inside the NC unit.

The expressions formed in this way are called codes, and this CNC system uses shift JIS codes.

The characters which are valid in this CNC system are listed below.

Character	Remarks
0 to 9	Always significant
A to Z	Always significant
+	Always significant
-	Always significant
.	Always significant
,	Always significant
/	Always significant
%	Always significant
CR	Always significant
LF/NL	Always significant
(Always significant
)	Always significant
:	Always significant
#	Always significant
*	Always significant
=	Always significant
[Always significant
]	Always significant
SP	Always significant
!	Always significant
\$	Always significant
BS	An error results during operation (except when the character is part of a comment).
HT	An error results during operation (except when the character is part of a comment).
&	An error results during operation (except when the character is part of a comment).
'(Apostrophe)	An error results during operation (except when the character is part of a comment).
;	An error results during operation (except when the character is part of a comment).
<	An error results during operation (except when the character is part of a comment).
>	An error results during operation (except when the character is part of a comment).
?	An error results during operation (except when the character is part of a comment).
@	An error results during operation (except when the character is part of a comment).
"	An error results during operation (except when the character is part of a comment).
DEL	Always ignored
NULL	Always ignored

2 Input Command

2.3 Program Format

2.3.2 Program Format

2.3.2.1 Format 1 for Lathe

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		–	–	–	–	–	–	–	–
	M system	–	–	–	–	–	–	–	–
	L system	○	○	○	○	○	○	○	○

The G-code of L system is selected by parameter.

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

2.3.2.3 Special Format for Lathe

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		–	–	–	–	–	–	–	–
	M system	–	–	–	–	–	–	–	–
	L system	○	○	○	○	○	○	○	○

2.3.2.4 Format 1 for Machining Center

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		○	○	○	○	○	○	○	○
	M system	○	○	○	○	○	○	○	○
	L system	–	–	–	–	–	–	–	–

2.3.2.7 Special Format for MELDAS

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		–	–	–	–	–	–	–	–
	M system	–	–	–	–	–	–	–	–
	L system	○	○	○	○	○	○	○	○

The formats of the turning fixed cycles (G77 to G79), multiple repetitive turning fixed cycles (G71 to G76) and drilling fixed cycles (G80 to G89) can be switched to the MELDAS special formats.

2 Input Command

2.4 Command Value

2.4 Command Value

2.4.1 Decimal Point Input I, II

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

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

(1) Decimal point input type I (When 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 unit.

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. Y200 F100 ^(*)	X100μm, F20mm/min Y200μm, F100mm/min	X100mm, F20mm/min Y200mm, F100mm/min
G4 X1.5	Dwell 1.5 s	←
X2	2ms	2s

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

2 Input Command

2.4 Command Value

2.4.2 Absolute / Incremental Command

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

(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)

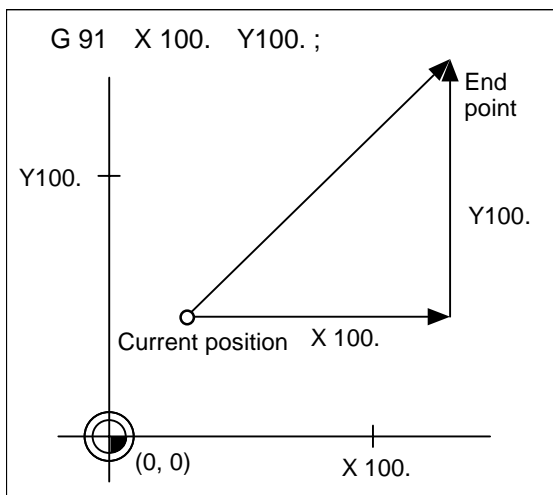
These G codes can be commanded multiple times in one block.

Example

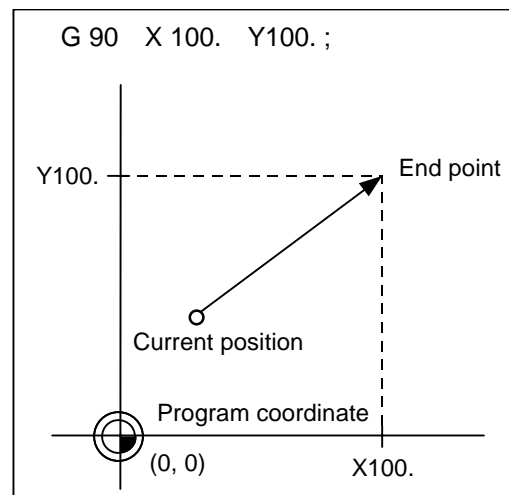
G90 X100.	G91 Y200.	G90 Z300.	;
Absolute value	Incremental value	Absolute value	

(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 Input Command

2.4 Command Value

(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.

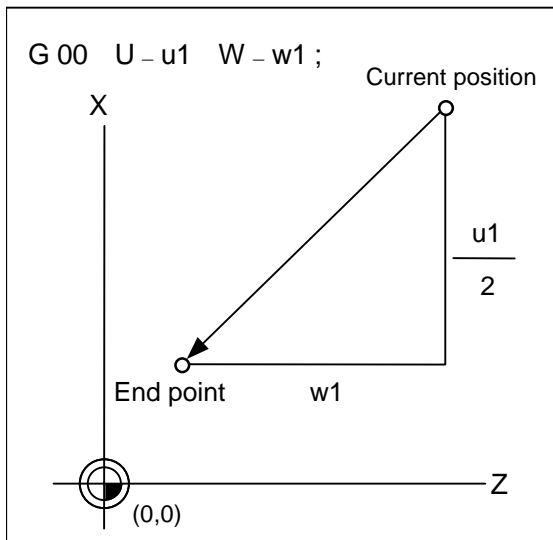
Absolute command (absolute value command) ... X, Z

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

Example

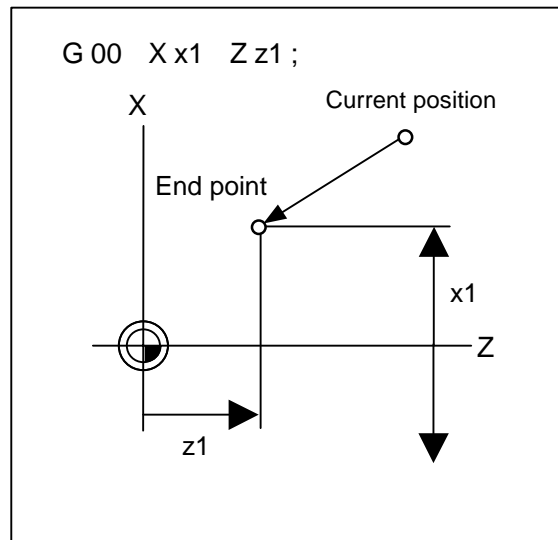
G00	X100.	W200.	;
	<u>Absolute value</u>	<u>Incremental value</u>	

(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) In addition to the above command method using axis addresses, to switch the absolute value command/incremental value command, the command method using G90/91 can also be selected using a parameter.

2 Input Command

2.4 Command Value

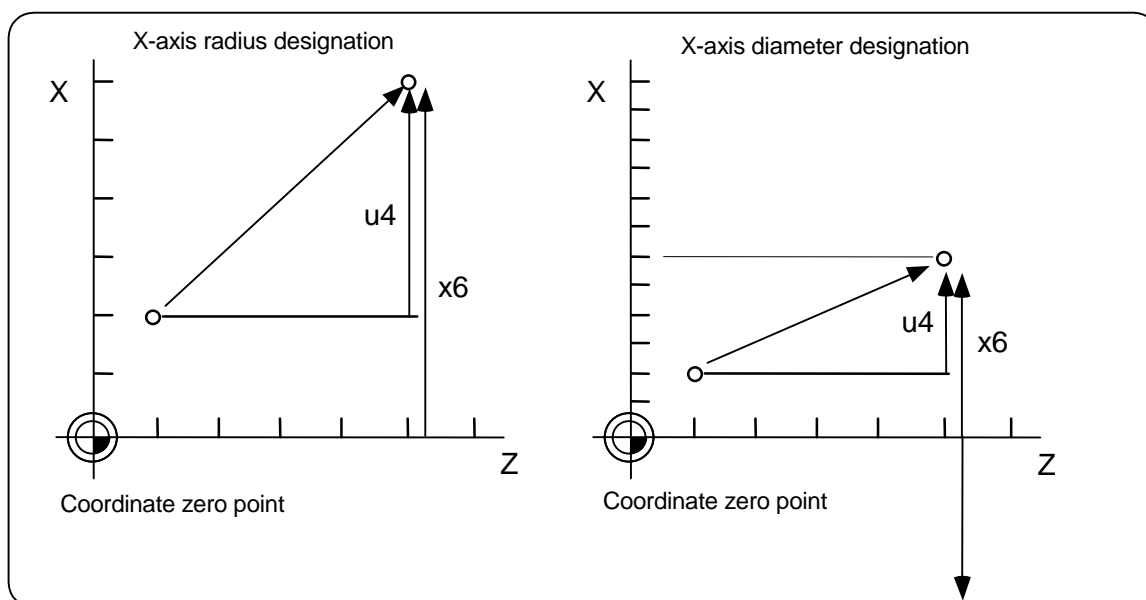
2.4.3 Diameter/Radius Designation

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	—	—	—	—	—	—	—	—
L system	○	○	○	○	○	○	○	○

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

When the diameter designation is selected, the scale of the length of the selected axis is doubled. (For instance, an actual length of 1 mm will be treated as 2 mm.)

This function is used when programming the work 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.



The difference in the diameter designation and radius designation is shown below.

Absolute value command		Incremental value command	
Radius designation	Diameter designation	Radius designation	Diameter designation
Actual movement amount = x1	Actual movement amount = 2 x1	Actual movement amount = u1	Actual movement amount = 2 u1

2 Input Command

2.5 Command Value and Setting Value Range

2.5 Command Value and Setting Value Range

2.5.1 Command Value and Setting Value Range

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

<Brief summary of format details>

[M system]

		Metric command	Inch command	Rotary axis (Metric command)	Rotary axis (Inch command)
Program number		08	←	←	←
Sequence number		N5	←	←	←
Preparatory function		G3/G21	←	←	←
Movement axis	0.001(°) mm/ 0.0001 inch	X+53 Y+53 Z+53 α+53	X+44 Y+44 Z+44 α+44	X+53 Y+53 Z+53 α+53	X+53 Y+53 Z+53 α+53
	0.0001(°) mm/ 0.00001 inch	X+44 Y+44 Z+44 α+44	X+35 Y+35 Z+35 α+35	X+44 Y+44 Z+44 α+44	X+44 Y+44 Z+44 α+44
Arc and cutter radius	0.001(°) mm/ 0.0001 inch	I+53 J+53 K+53 R+53	I+44 J+44 K+44 R+44	I+53 J+53 K+53 R+53	I+44 J+44 K+44 R+44 (Note 5)
	0.0001(°) mm/ 0.00001 inch	I+44 J+44 K+44 R+44	I+35 J+35 K+35 R+35	I+44 J+44 K+44 R+44	I+35 J+35 K+35 R+35 (Note 5)
Dwell	0.001(°) mm/ 0.0001 inch	X+53/P+8	←	←	←
	0.0001(°) mm/ 0.00001 inch	X+53/P+8	←	←	←
Feed function	0.001(°) mm/ 0.0001 inch	F63(Feed per minute) F43(Feed per revolution)	F44(Feed per minute) F34(Feed per revolution)	F63(Feed per minute) F43(Feed per revolution)	F44(Feed per minute) F34(Feed per revolution) (Note 6)
	0.0001 (°) mm/ 0.00001 inch	F54(Feed per minute) F34(Feed per revolution)	F35(Feed per minute) F25(Feed per revolution)	F54(Feed per minute) F34(Feed per revolution)	F35(Feed per minute) F25(Feed per revolution) (Note 6)
Tool offset		H3 D3	←	←	←
Miscellaneous function (M)		M8	←	←	←
Spindle function (S)		S8	←	←	←
Tool function (T)		T8	←	←	←
2nd miscellaneous function		A8/B8/C8	←	←	←
Subprogram		P8 H5 L4	←	←	←
Fixed cycle	0.001(°) mm/ 0.0001 inch	R+53 Q53 P8 L4	←	←	←
	0.0001(°) mm/ 0.00001 inch	R+44 Q44 P8 L4	←	←	←

2 Input Command

2.5 Command Value and Setting Value Range

[L system]

		Metric command	Inch command	Rotary axis (Metric command)	Rotary axis (Inch command)
Program number		08	←	←	←
Sequence number		N5	←	←	←
Preparatory function		G3/G21	←	←	←
Movement axis	0.001(°) mm/ 0.0001 inch	X+53 Z+53 α+53	X+44 Z+44 α+44	X+53 Z+53 α+53	X+53 Z+53 α+53
	0.0001(°) mm/ 0.00001 inch	X+44 Z+44 α+44	X+35 Z+35 α+35	X+44 Z+44 α+44	X+44 Z+44 α+44
	0.00001(°) mm/ 0.000001 inch	X+35 Z+35 α+35	—	X+35 Z+35 α+35	—
Arc and cutter radius	0.001(°) mm/ 0.0001 inch	I+53 K+53 R+53	I+44 K+44 R+44	I+53 K+53 R+53	I+44 K+44 R+44 (Note 5)
	0.0001(°) mm/ 0.00001 inch	I+44K+44 R+44	I+35 K+35 R+35	I+44 K+44 R+44	I+35 K+35 R+35 (Note 5)
	0.00001(°) mm/ 0.000001 inch	I+35 K+35 R+35	—	I+35 K+35 R+35	—
Dwell	0.001(°) mm/ 0.0001 inch	X+53/P+8	←	←	←
	0.0001(°) mm/ 0.00001 inch	X+53/P+8	←	←	←
	0.00001(°) mm/ 0.000001 inch	X+53/P+8	—	X+53/P+8	—
Feed function	0.001(°) mm/ 0.0001 inch	F63(Feed per minute) F43(Feed per revolution)	F44(Feed per minute) F34(Feed per revolution)	F63(Feed per minute) F44(Feed per revolution)	F44(Feed per minute) F34(Feed per revolution) (Note 6)
	0.0001(°) mm/ 0.00001 inch	F54(Feed per minute) F34(Feed per revolution)	F35(Feed per minute) F25(Feed per revolution)	F54(Feed per minute) F34(Feed per revolution)	F35(Feed per minute) F25(Feed per revolution) (Note 6)
	0.00001(°) mm/ 0.000001 inch	F45(Feed per minute) F25(Feed per revolution)	—	F45(Feed per minute) F25(Feed per revolution)	—
Tool offset		T1/T2	←	←	←
Miscellaneous function (M)		M8	←	←	←
Spindle function (S)		S8	←	←	←
Tool function (T)		T8	←	←	←
2nd miscellaneous function		A8/B8/C8	←	←	←
Subprogram		P8 H5 L4	←	←	←
Fixed cycle	0.001(°) mm/ 0.0001 inch	R+53 Q53 P8 L4	←	←	←
	0.0001(°) mm/ 0.00001 inch	R+44 Q44 P8 L4	←	←	←
	0.00001(°) mm/ 0.000001 inch	R+35 Q35 P8 L4	—	R+35 Q35 P8 L4	—

(Note 1) α indicates the additional axis address, such as A, B or C.

(Note 2) The No. of digits check for a word is carried out with the maximum number of digits of that address.

(Note 3) Numerals can be used without the leading zeros.

(Note 4) The meanings of the details are as follows :

Example 1 : 08 : 8-digit program number

Example 2 : G21 : Dimension G is 2 digits to the left of the decimal point, and 1 digit to the right.

Example 3 : X+53 : Dimension X uses + or - sign and represents 5 digits to the left of the decimal point and 3 digits to the right.

For example, the case for when the X axis is positioned (G00) to the 45.123 mm position in the absolute value (G90) mode is as follows :

G00 X45.123 ;

3 digits below the decimal point
 5 digits above the decimal point, so it's +00045, but the leading zeros and the mark (+) have been omitted.
 G0 is possible, too.

2 Input Command

2.5 Command Value and Setting Value Range

- (Note 5)** If an arc is commanded using a rotary axis and linear axis while inch commands are being used, the degrees will be converted into 0.1 inches for interpolation.
- (Note 6)** While inch commands are being used, the rotary axis speed will be in increments of 10 degrees.
Example : With the F1. (per-minute-feed) command, this will become the 10 degrees/minute command.
- (Note 7)** The decimal places below the decimal point are ignored when a command, such as an S command, with an invalid decimal point has been assigned with a decimal point.
- (Note 8)** This format is the same for the value input from the memory, MDI or setting and display unit.
- (Note 9)** Command the program No. in an independent block. Command the program NO. in the head block of the program.

2 Input Command

2.5 Command Value and Setting Value Range

<List of Command Value and Setting Value Ranges>

	Linear axis		Rotary axis
	Input unit: mm	Input unit: inch	Degree (°)
Minimum input setting unit	0.001/0.0001/0.00001	0.0001/0.00001	0.001/0.0001/0.00001
Maximum stroke (Value on machine coordinate system)	±99999.999 mm ±9999.9999 mm ±999.99999 mm	±9999.9999 inch ±999.99999 inch	±99999.999 ° ±9999.9999 ° ±999.99999 °
Maximum command value	±99999.999 mm ±9999.9999 mm ±999.99999 mm	±9999.9999 inch ±999.99999 inch	±99999.999 ° ±9999.9999 ° ±999.99999 °
Rapid traverse rate (Including during dry run)	1 to 1000000 mm/min 1 to 100000 mm/min 1 to 10000 mm/min	1 to 39370 inch/min 1 to 3937 inch/min	1 to 1000000 °/min 1 to 100000 °/min 1 to 10000 °/min
M system cutting feed rate (Including during dry run)	0.01 to 1000000 mm/min 0.001 to 100000 mm/min -	0.001 to 100000 inch/min 0.0001 to 10000 inch/min	0.01 to 1000000 °/min 0.001 to 100000 °/min -
L system cutting feed rate (Including during dry run)	0.001 to 1000000 mm/min 0.0001 to 100000 mm/min 0.00001 to 10000 mm/min	0.0001 to 39370.0787 inch/min 0.00001 to 3937.00787 inch/min	0.001 to 1000000 °/min 0.0001 to 100000 °/min 0.00001 to 10000 °/min
M system synchronous feed	0.001 to 999.999 mm/rev 0.0001 to 99.9999 mm/rev -	0.0001 to 999.9999 inch/rev 0.00001 to 99.99999 inch/rev	0.01 to 999.99 °/rev 0.001 to 99.999 °/rev -
L system synchronous feed	0.0001 to 999.9999 mm/rev 0.00001 to 99.99999 mm/rev 0.000001 to 9.999999 mm/rev	0.000001 to 99.999999 inch/rev 0.0000001 to 9.9999999 inch/rev	0.0001 to 999.9999 °/rev 0.00001 to 99.99999 °/rev 0.000001 to 9.999999 °/rev
2nd to 4th reference point offset (value on machine coordinate system)	±99999.999 mm ±9999.9999 mm ±999.99999 mm	±9999.9999 inch ±999.99999 inch	±99999.999 ° ±9999.9999 ° ±999.99999 °
Tool offset amount (shape)	±999.999 mm ±99.9999 mm ±9.99999 mm	±99.9999 inch ±9.99999 inch	
Tool offset amount (wear)	±9999.999 mm ±999.9999 mm ±99.99999 mm	±9.9999 inch ±0.99999 inch	
Incremental feed amount	0.001 mm/pulse 0.0001 mm/pulse 0.00001 mm/pulse	0.0001 inch/pulse 0.00001 inch/pulse	0.001 °/pulse 0.0001 °/pulse 0.00001 °/pulse
Handle feed amount	0.001 mm/pulse 0.0001 mm/pulse 0.00001 mm/pulse	0.0001 inch/pulse 0.00001 inch/pulse	0.001 °/pulse 0.0001 °/pulse 0.00001 °/pulse
Soft limit range (value on machine coordinate system)	-99999.999 mm to +99999.999 mm -9999.9999 mm to +9999.9999 mm -999.99999 mm to +999.99999 mm	-9999.9999 inch to +9999.9999 inch -999.99999 inch to +999.99999 inch	1 to 359.999 ° 1 to 359.9999 ° 1 to 359.99999 °
Dwell time	0 to 99999.999 s	0 to 99999.999 s	
Backlash compensation amount	0 to ±9999 pulse	0 to ±9999 pulse	0 to ±9999 pulse
Pitch error compensation	0 to ±9999 pulse	0 to ±9999 pulse	0 to ±9999 pulse
M system thread lead (F)	0.001 to 999.999 mm/rev 0.0001 to 99.9999 mm/rev -	0.0001 to 99.9999 inch/rev 0.00001 to 9.99999 inch/rev	
M system thread lead (Precise E)	0.00001 to 999.99999 mm/rev 0.000001 to 99.999999 mm/rev -	0.000001 to 39.370078 inch/rev 0.000001 to 3.937007 inch/rev	
L system thread lead (F)	0.0001 to 999.9999 mm/rev 0.00001 to 99.99999 mm/rev 0.000001 to 9.999999 mm/rev	0.000001 to 99.999999 inch/rev 0.0000001 to 9.9999999 inch/rev	
L system thread lead (Precise E)	0.00001 to 999.99999 mm/rev 0.000001 to 99.999999 mm/rev 0.0000001 to 99.9999999 mm/rev	0.000010 to 9.9999999 inch/rev 0.0000010 to 0.99999999 inch/rev	

(Note 1) The second line in the table applies when the least setting increment is 0.001, 0.0001 from the first line. The third line applies when the least setting increment is 0.001, 0.0001, 0.00001 from the first line.

3 Positioning / Interpolation

3.1 Positioning

3 Positioning / Interpolation

3.1 Positioning

3.1.1 Positioning

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

This function carries out positioning at high speed using rapid traverse 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.

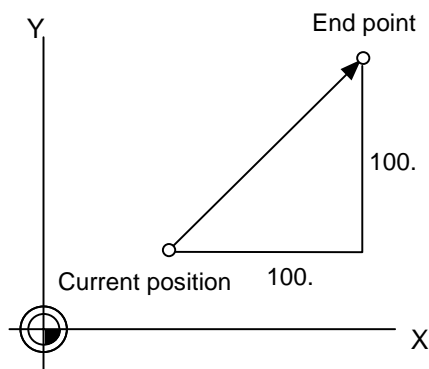
For details on the rapid traverse feed rate of the NC, refer to the section entitled "Rapid Traverse Rate". Since the actual rapid traverse feed rate depends on the machine, refer to the specifications of the machine concerned.

- (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)

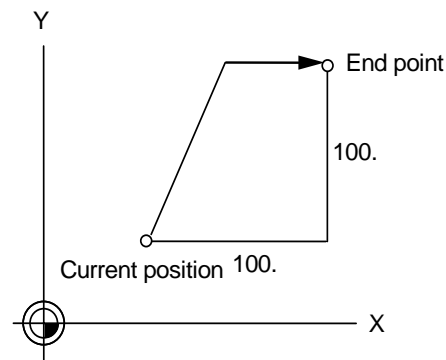
(Example) Linear type (Moves linearly to the end point.)

G 00 G 91 X 100. Y 100. ;



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

G 00 G 91 X 100. Y 100. ;



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

3 Positioning / Interpolation

3.1 Positioning

3.1.2 Unidirectional Positioning

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	-	-	-	-	-	-	-	-

The G60 command always moves the tool to the final position in a direction determined with parameters.

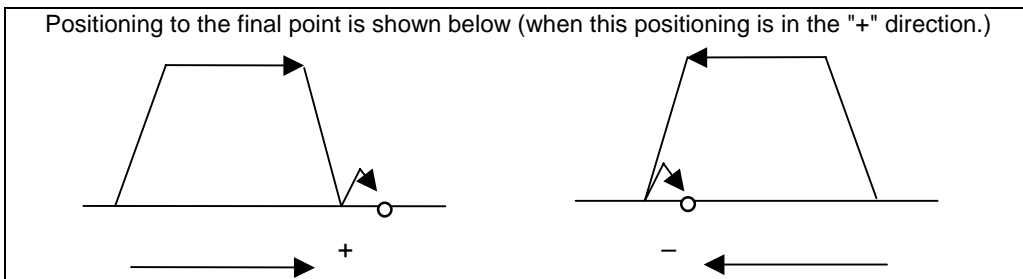
The tool can be positioned without backlash.

G60 Xx1 Yy1 Zz1 ; (Also possible for additional axes A, B, C, U, V, W simultaneously)

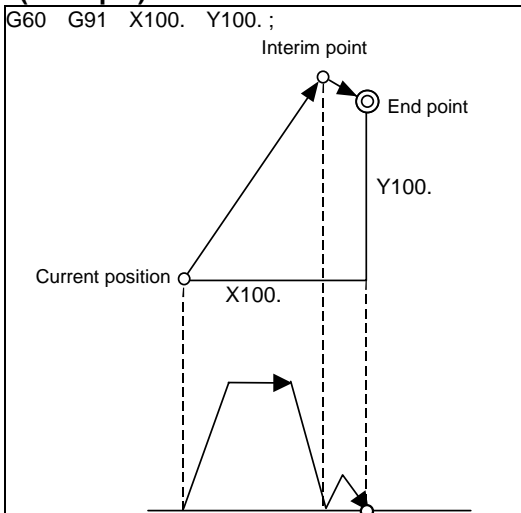
(x1, y1, z1: numerical values denoting the position data)

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".

3 Positioning/Interpolation

3.2 Linear/Circular Interpolation

- (Note 1)** The processing of the above pattern will be followed even for the machine lock and Z-axis command cancel.
- (Note 2)** The creep distance 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/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

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

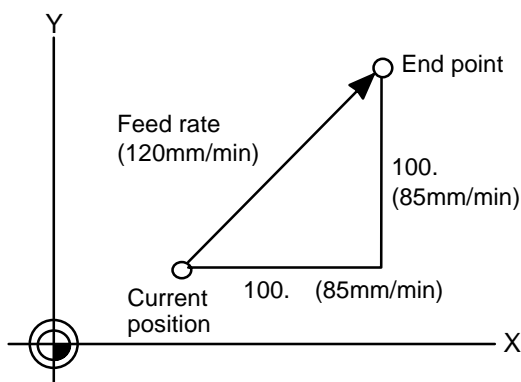
G01 Xx1 Yy1 Zz1 Ff1 ; (Also possible for additional axes A, B, C, U, V, W simultaneously)

(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 Positioning/Interpolation

3.2 Linear/Circular Interpolation

3.2.2 Circular Interpolation (Center / Radius Designation)

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

(1) Circular interpolation with I, J, K commands

This function moves a tool along a circular arc on the plane selected by the plane selection G code with movement command 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 values
 Ff1 : Feed rate

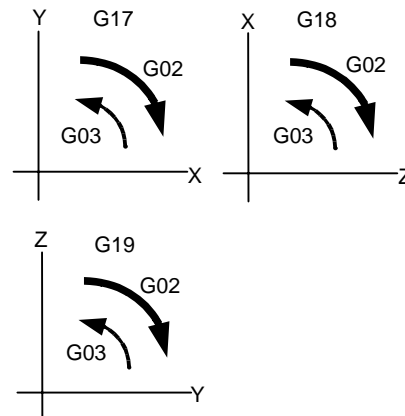
The above commands move the tool along the circular arc at the f1 feed rate. The tool moves along a circular path, whose center is the position from the start point designated by distance "i1" in the X-axis direction and distance "j1" in the Y-axis direction, toward the end point.

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

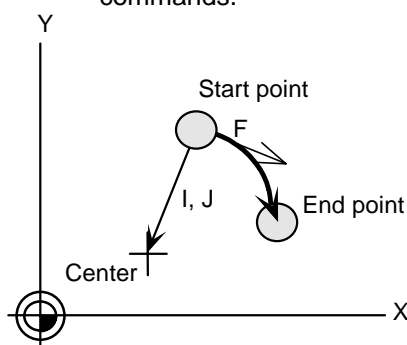
- G02: Clockwise (CW)
- G03: Counterclockwise (CCW)

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".

3 Positioning/Interpolation

3.2 Linear/Circular Interpolation

(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

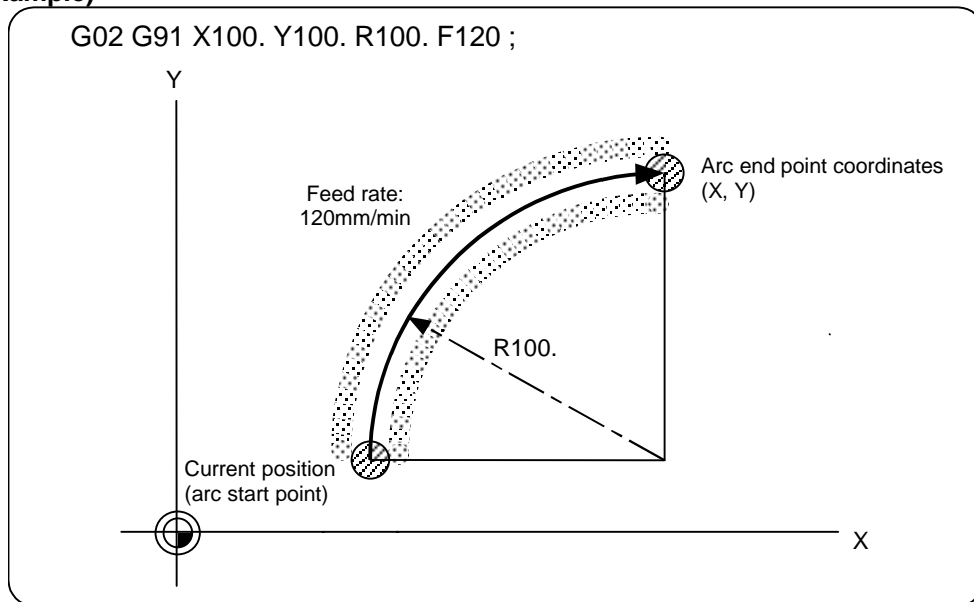
G02 or G03 is used to designate the direction of the arc rotation.

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 Positioning/Interpolation

3.2 Linear/Circular Interpolation

3.2.3 Helical Interpolation

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	–	–	–	–	☆	☆	☆	☆

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

G17 G02(G03) Xx1 Yy1 Zz1 Ii1 Jj1 Pp1 Ff1 ; G17 : Arc plane G02, G03 : Arc rotation direction Xx1, Yy1 : End point coordinate values for arc Zz1 : End point coordinate of linear axis Ii1, Jj1 : Arc center coordinates Pp1 : Pitch No. Ff1 : Feed rate

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

Pitch I1 is obtained by the formula below.

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

$$\theta = \theta_e - \theta_s = \arctan(y_e/x_e) - \arctan(y_s/x_s)$$

Where x_s, y_s are the start point coordinates ($0 \leq \theta < 2\pi$)
 x_e, y_e are the end point coordinates

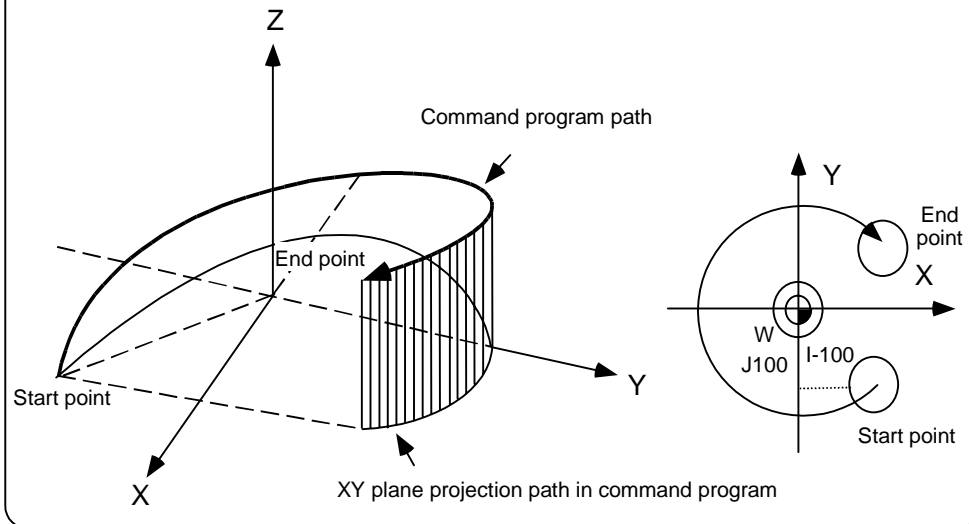
The combination of the axes which can be commanded simultaneously depends on the specifications. The axes can be used in any combination within this range. The feed rate is controlled so that the tool always moves at a speed along the circumference of the circle.

3 Positioning/Interpolation

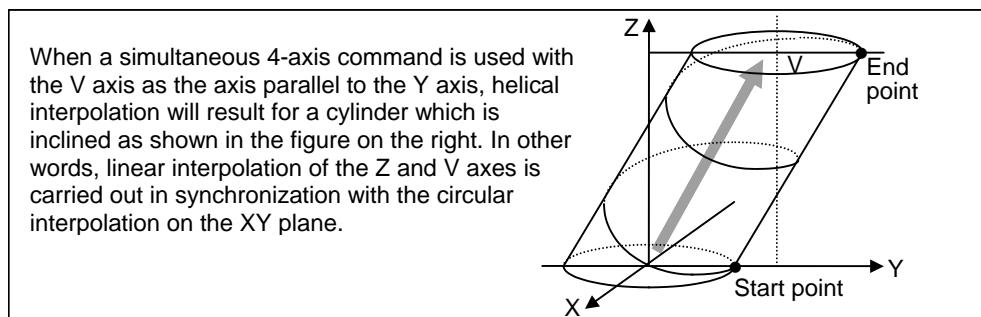
3.2 Linear/Circular Interpolation

(Example)

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



(Note 1) Helical sharps 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.



3 Positioning/Interpolation

3.2 Linear/Circular Interpolation

3.2.4 Spiral / Conical Interpolation

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	-	Δ	Δ
L system	-	-	-	-	-	-	-	-

This function interpolates arcs where the start point and endpoint are not on the circumference of the same circle into spiral shapes.

G17 G02.1(G03.1) Xx1 Yy1 Ii1 Jj1 Pp1 Ff1 ;
G17 : Arc plane G02.1, G03.1 : Arc rotation direction Xx1, Yy1 : End point coordinate Ii1, Jj1 : Arc center value Pp1 : Pitch number Ff1 : Feed rate

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

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

- G17: XY plane
- G18: ZX plane
- G19: YZ plane

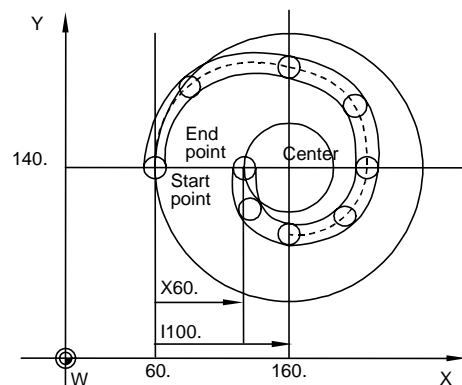
The direction of the arc rotation is designated by G2.1 or G3.1.

- G2.1: Clockwise (CW)
- G3.1: Counterclockwise (CCW)

The pitch number (number of rotations) is designated by p1.

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

Example: G91 G17 G01 X60. F500 ;
 Y140. ;
 G2.1 X60. Y0 I100. P1 F300 ;
 G01 X-120 ;
 G90
 G17 G01 X60. F500 ;
 Y140. ;
 G2.1 X120. Y140. I100. P1 F300 ;
 G01 X0 ;



3 Positioning/Interpolation

3.2 Linear/Circular Interpolation

(1) The combination of the axes which can be commanded simultaneously depends on the specifications. Any combination can be used within the specified range.

(2) The feed rate is a constant tangential rate.

(Note 1) This function cannot be used in combination with a tool radius compensation command (G41, G42).

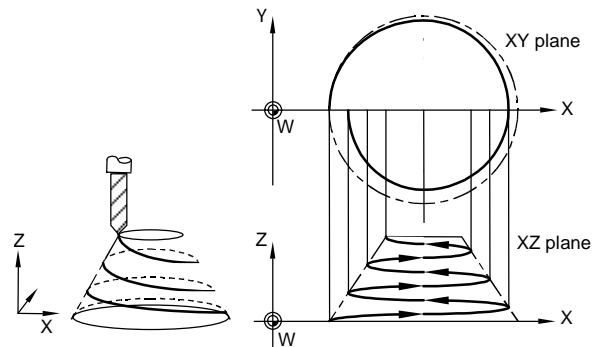
(Note 2) The arc plane is always based on the G17, G18 or G19 command. Arc control is performed on a plane by the G17, G18 or G19 command even when two addresses which are not on any one plane are designated.

Example:

When an axis other than the ones for the spiral interpolation plane has been designated at the same time, the other axis will also be interpolated in synchronization with the spiral interpolation.

G17 G91 G2.1 X100. Z150. I150.
P3 F500 ;

In the example given above, truncated cone interpolation is performed.



In the case of command format type 2
Spiral interpolation

G17 G02(G03) Xx1 Yy1 Ii1 Jj1 Qq1/LI1 Ff1 ;
G17 : Arc plane
G02, G03 : Arc rotation direction
Xx1, Yy1 : End point coordinate
Ii1, Jj1 : Arc center values
Qq1 : Incremental/decremental amount of radius per spiral rotation
LI1 : Pitch number
Ff1 : Feed rate

- Relation between Q and L
 $L = |(\text{arc end point radius} - \text{arc start point radius})| / |Q|$
- Q takes precedence if both Q and L have been designated at the same time.

Conical interpolation

G17 G02(G03) Xx1 Yy1 Zz1 Ii1 Jj1 Kk1/Qq/LI1 Ff1 ;
Zz1 : End point coordinate in height direction
Kk1 : Amount by which height is incremented or decremented per spiral rotation
Qq1 : Amount by which radius is incremented or decremented per spiral rotation
LI1 : Pitch number
Ff1 : Feed rate

- Relation between L and (I, J) K
 $L = | \text{Height} | / | \text{Amount by which height is incremented or decremented (I, J, K)} |$
- Q takes precedence over K which in turn takes precedence over L if Q, K and L have been designated at the same time.

3 Positioning/Interpolation

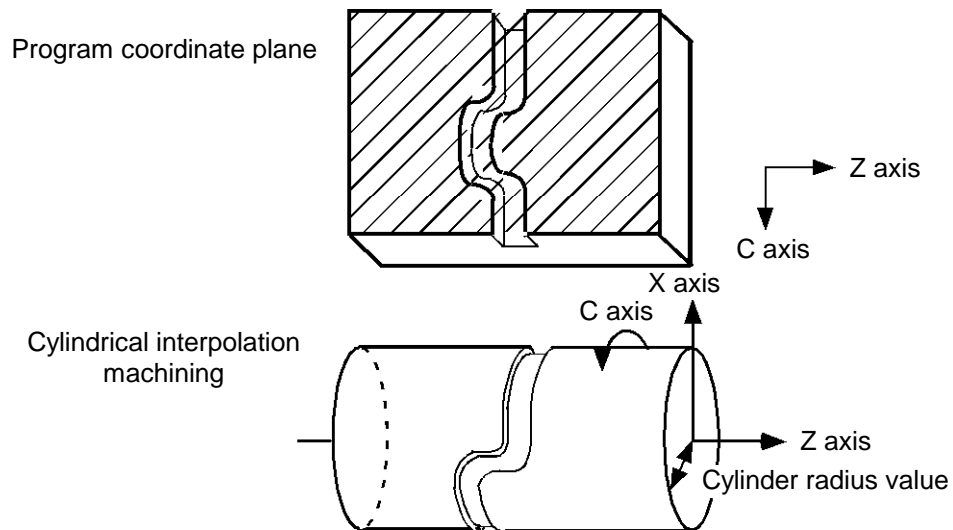
3.2 Linear/Circular Interpolation

3.2.5 Cylindrical Interpolation

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	Δ	Δ	Δ
L system	-	-	-	-	-	Δ	Δ	Δ

This function transfers the shape that is on the cylinder's side surface (shape yielded by the cylindrical coordinate system) onto a plane, and when the transferred shape is designated in the program in the form of plane coordinates, the shape is converted into a movement along the linear and rotary axes of the original cylinder coordinates, and the contours are controlled by means of the CNC unit during machining.

Since the programming can be performed for the shapes produced by transferring the side surfaces of the cylinders, this function is useful when it comes to machining cylindrical cams and other such parts.



- (1) Cylindrical interpolation mode start
(G07.1 name of rotary axis cylinder radius value;)

Cylindrical interpolation is performed between the rotary axis designated in the G07.1 block and any other linear axis.

- (a) Linear interpolation or circular interpolation can be designated in the cylindrical interpolation mode. However, assign the G19 command (plane selection command) immediately before the G07.1 block.
- (b) The coordinates can be designated either in absolute values or incremental values.
- (c) Tool radius compensation can be applied to the program commands. Cylindrical interpolation is performed for the path after tool radius compensation.
- (d) For the feed rate, designate a tangential rate over the cylinder transfer surface using the F command.

The F rate is in either mm/min or inch/mm units.

- (2) Cylindrical interpolation mode cancel
(G07.1 name of rotary axis 0;)

If "C" is the name of the rotary axis, the cylindrical interpolation cancel mode is established with the command below.

G07.1 C0 ;

3 Positioning/Interpolation

3.2 Linear/Circular Interpolation

3.2.6 Polar Coordinate Interpolation

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	-	Δ	Δ
L system	-	-	-	-	-	Δ	Δ	Δ

This function converts the commands programmed by the orthogonal coordinate axes into rectilinear axis movements (tool movements) and rotary axis movements (workpiece rotation) to control the contours. It is useful for cutting linear cutouts on the outside diameter of the workpiece, grinding cam shafts, etc.

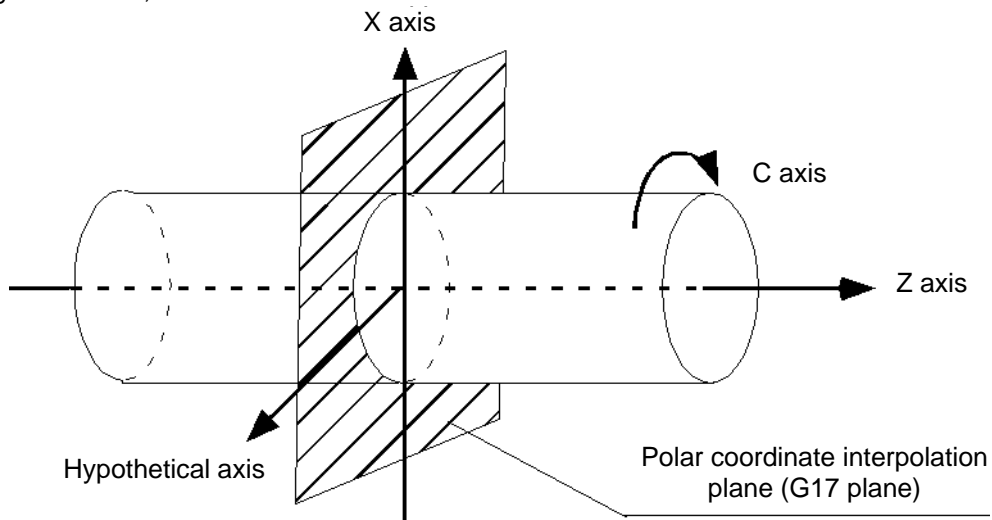


Fig. 1 Diagram explaining polar coordinate interpolation axes

(1) Polar coordinate interpolation mode
(G12.1)

The polar coordinate interpolation mode is established by designating the G12.1 command. The plane (hereafter referred to as the "polar coordinate interpolation plane"), for which the linear axis will be first orthogonal axis and the hypothetical axis at right angles to the linear axis will be the second axis, is selected. Polar coordinate interpolation is performed on this plane.

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

(2) Polar coordinate interpolation cancel mode
(G13.1)

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

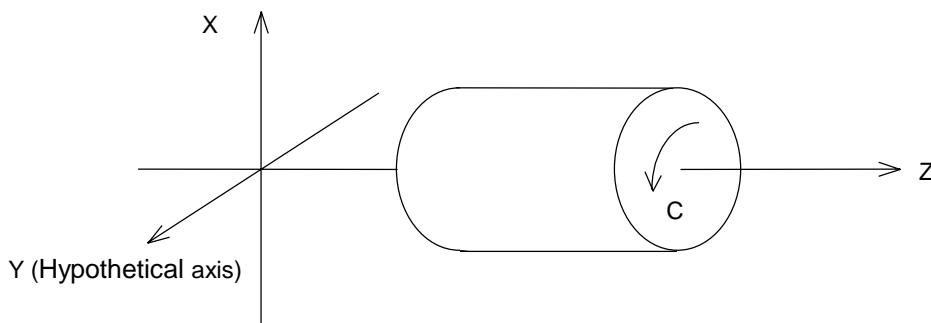
3 Positioning/Interpolation

3.2 Linear/Circular Interpolation

3.2.7 Milling Interpolation

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	-	-	-
L system	-	-	-	-	-	-	Δ	Δ

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



G12.1 ; Milling mode ON
 G13.1 ; Milling mode OFF (Turning mode)

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

3 Positioning/Interpolation

3.3 Curve Interpolation

3.3 Curve Interpolation

3.3.2 Exponential Interpolation

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	-	Δ	Δ
L system	-	-	-	-	-	-	Δ	Δ

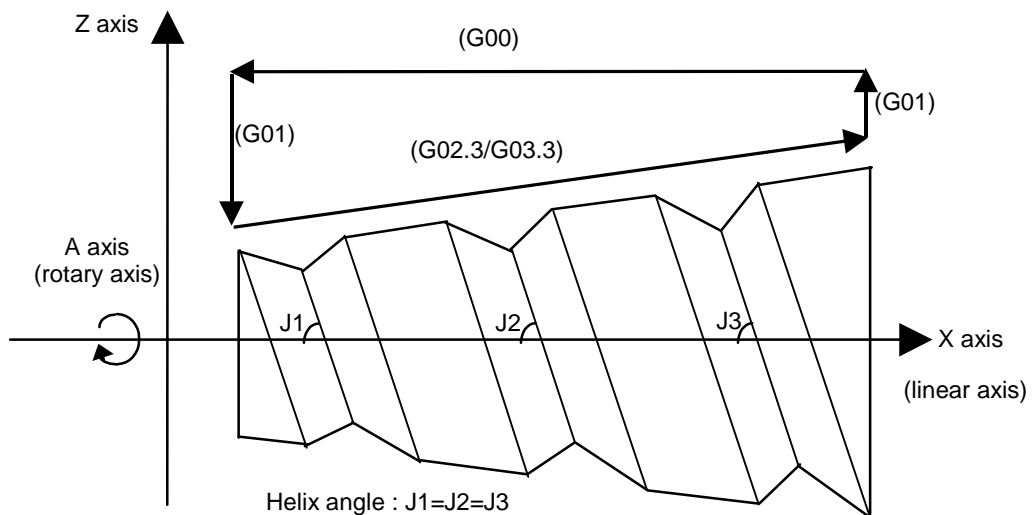
With this function, the rotary axis is changed into exponential functions vis-a-vis the linear axis movements to achieve interpolation.

When exponential function interpolation is performed, linear interpolation is performed between the other axes and the linear axis.

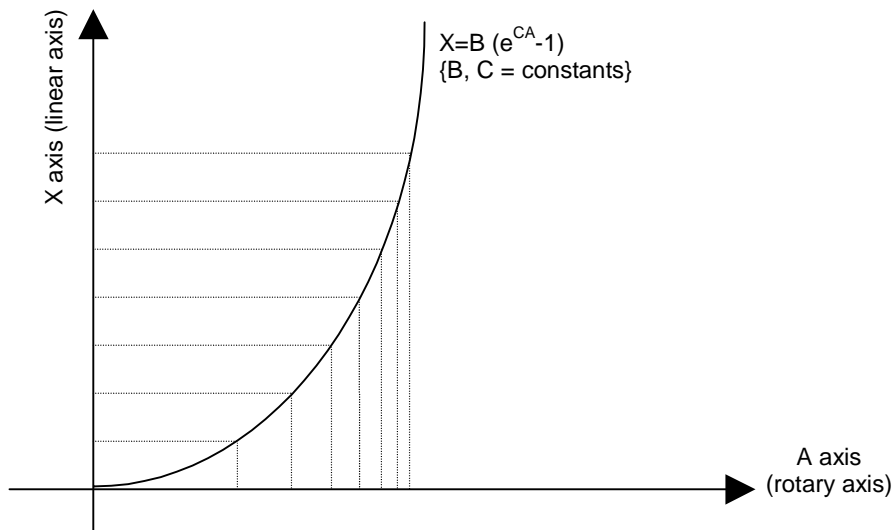
This makes it possible to machine tapered grooves (equal helix machining of tapered shapes) whose helix angle is always constant.

The function can be used for slotting and grinding end mills and other tools.

[Equal helix machining of tapered shapes]



[Relationship between linear and rotary axes]



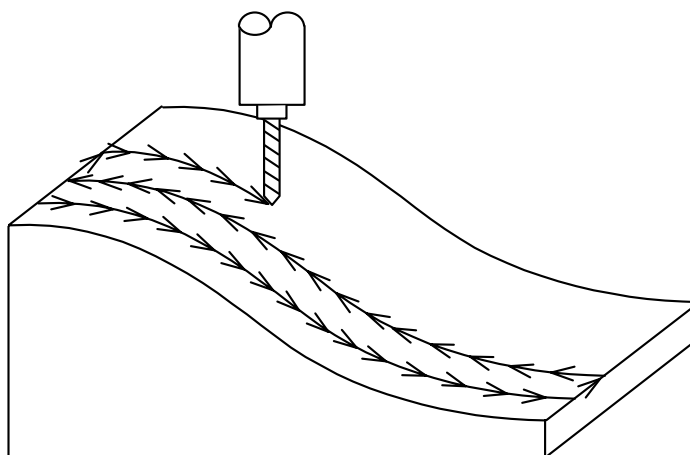
3 Positioning/Interpolation

3.3 Curve Interpolation

3.3.3 Spline Interpolation

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	-	Δ	Δ
L system	-	-	-	-	-	-	-	-

This function automatically generates spline curves that smoothly pass through rows of dots designated by a fine-segment machining program, and performs interpolation for the paths along the curves. This enables high-speed and high-accuracy machining to be achieved. To use this function, the high-accuracy control function (G08P1) is required.



4 Feed

4.1 Feed Rate

4 Feed

4.1 Feed Rate

4.1.1 Rapid Traverse Rate (m/min)

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	1000	1000	1000	1000	1000	1000	1000	1000
L system	1000	1000	1000	1000	1000	1000	1000	1000

[M system]

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

G00, G27, G28, G29, G30 and G60 are the effective commands for the rapid traverse rate.

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.

G00, G27, G28, G29, G30 and G53 are the effective commands for the rapid traverse rate.

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

- Rapid Traverse Rate setting range

Least input increment	B	C	D
Metric input	1~1000000 (mm/min, °/min)	1~100000 (mm/min, °/min)	1~10000 (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)

Least input increment D : 0.00001 mm

4 Feed

4.1 Feed Rate

4.1.2 Cutting Feed Rate (m/min)

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	1000	1000	1000	1000	1000	1000	1000	1000
L system	1000	1000	1000	1000	1000	1000	1000	1000

[M system]

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

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

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

- Cutting Feed Rate setting range

Least 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)

- Effective G-code commands for the cutting feed
G01, G02, G03, G2.1 G3.1 G33, 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	D
Metric input	1~1000000 (mm/min, °/min)	1~100000 (mm/min, °/min)	1~10000 (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)

Least input increment D : 0.00001 mm

- Effective G-code commands for the cutting feed
G01, G02, G03, G33, etc. As to others, refer to interpolation specifications.

4 Feed

4.1 Feed Rate

4.1.3 Manual Feed Rate (m/min)

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	1000	1000	1000	1000	1000	1000	1000	1000
L system	1000	1000	1000	1000	1000	1000	1000	1000

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

The manual feed rate signals from the PLC 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 numerical value 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 numerical value setting method.

4 Feed

4.2 Feed Rate Input Methods

4.2 Feed Rate Input Methods

4.2.1 Feed per Minute

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

[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	F1 = 1 mm/min	F1 = 1 mm/min
	with decimal point	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	F1 = 1 inch/min	F1 = 1 inch/min
	with decimal point	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 (I_Sync) to OFF.
- The F command increments are common to the entire part system.

4 Feed

4.2 Feed Rate Input Methods

[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	(D) 0.00001 mm
F command increment (mm/min)	without decimal point	F1 = 1 mm/min F1. = 1 mm/min	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	0.00001 ~10000.00000

Inch input (inch)

Least input increment		(B) 0.0001 inch	(C) 0.00001 inch
F command increment (inch/min)	without 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 (I_Sync) to OFF.

4 Feed

4.2 Feed Rate Input Methods

4.2.2 Feed per Revolution

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	○	○	○	○

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

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

[M system]

Metric input (mm)

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 (I_Sync) to OFF.
- The F command increments are common to the entire part system.

[L system]

Metric input (mm)

Least input increment		(B) 0.001 mm	(C) 0.0001 mm	(D) 0.00001 mm
F command increment (mm/rev)	without decimal point	F1 = 0.0001	F1 = 0.0001	F1 = 0.0001
	with decimal point	F1. = 1	F1. = 1	F1. = 1
Command range (mm/rev)		0.0001~999.999	0.00001~99.99999	0.000001~9.999999

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 (I_Sync) to OFF.

4 Feed

4.2 Feed Rate Input Methods

4.2.4 F 1-digit Feed

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

When the "F1digit" 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 established when F0 is designated which is the same as the G00 command. When one of the codes F1 to F5 is designated, the cutting feed rate set to support the code serves as the valid rate command. When a command higher than F5 is designated, it serves as a regular direct command with feed rate value of 5 digits following address F.

When an F 1-digit command has been designated, the "In F 1-digit" external output signal is output.

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

(Note) When using M64 and M64A, the feed rate cannot be increased or decreased using the manual handle.

- (1) Amount by which speed is varied by manual handle
Speed variation amount ΔF is expressed by the equation below:

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

ΔP : Handle pulses (\pm)
 FM : F1 to F5 upper limit (parameter setting)
 K : Speed variation constant (parameter setting)

Example: When the feed rate is to be increased or reduced by 10 mm/min per manual handle scale increment

If F_{max} is 3600 mm/min, then: $\Delta F = 10 = 1 \times \frac{3600}{K}$

Therefore, $K = 360$.

- (2) Conditions under which F1-digit feed is valid
- (a) The automatic operation must be selected.
 - (b) Automatic start must be underway.
 - (c) Cutting feed must be underway, and the F 1-digit feed rate must be designated.
 - (d) The F 1-digit valid parameter must be ON.
 - (e) The F 1-digit feed rate change valid signal must be ON.
 - (f) A dry run must not be in progress.

4 Feed

4.3 Override

4.3 Override

4.3.1 Rapid Traverse Override

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
	M system	○	○	○	○	○	○	○	○
	L system	○	○	○	○	○	○	○	○

(1) Type 1 (code method)

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

(2) Type 2 (value setting method)

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

(Note 1) Type 1 and type 2 can be selected by PLC processing.

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

4.3.2 Cutting Feed Override

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
	M system	○	○	○	○	○	○	○	○
	L system	○	○	○	○	○	○	○	○

(1) Type 1 (code method)

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

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

(2) Type 2 (value setting method)

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

(Note 1) A PLC must be built into the unit for type 2.

4 Feed

4.3 Override

4.3.3 2nd Cutting Feed Override

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

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

(Note 1) A PLC must be built into the unit for this function.

4.3.4 Override Cancel

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

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

(Note 1) The override cancel signal is not valid for manual operation.

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

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

4 Feed

4.4 Acceleration / Deceleration

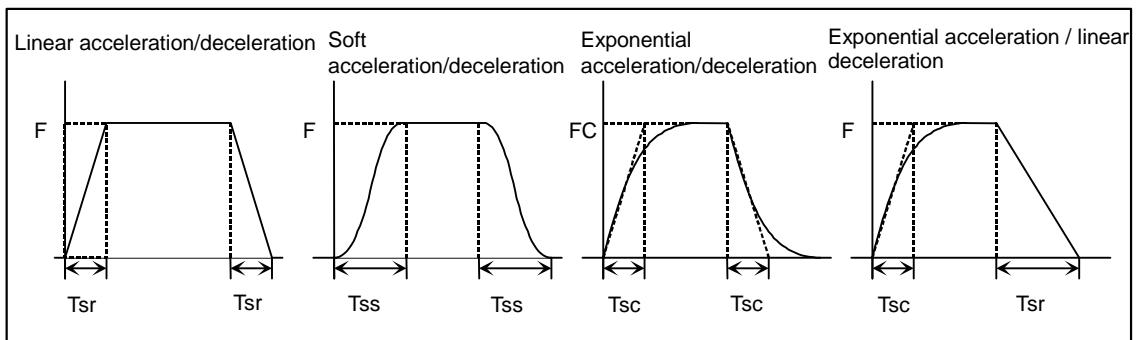
4.4 Acceleration / Deceleration

4.4.1 Automatic Acceleration / Deceleration after Interpolation

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

Acceleration/deceleration is applied to all commands automatically. The acceleration/deceleration patterns are linear acceleration/deceleration, soft acceleration/deceleration, exponent function acceleration/deceleration, exponent function acceleration/linear deceleration and any of which can be selected by using a parameter.

For rapid traverse feed or manual feed, acceleration/deceleration is always made for each block, and the time constant can be set for each axis separately.



(Note 1) The rapid traverse feed acceleration/deceleration patterns are also effective for the following:

G00, G27, G28, G29, G30, rapid traverse feed in manual run, JOG, incremental feed, return to reference position.

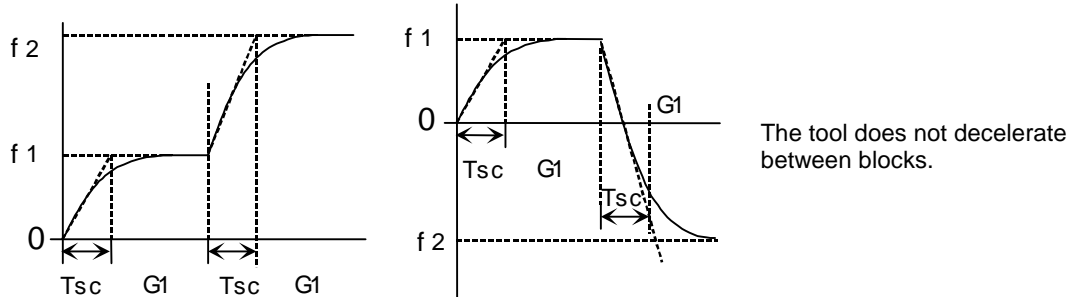
(Note 2) Acceleration/deceleration in handle feed mode is usually performed according to the acceleration/deceleration pattern for cutting feed. However, a parameter can be specified to select a pattern with no acceleration/deceleration (step).

4 Feed

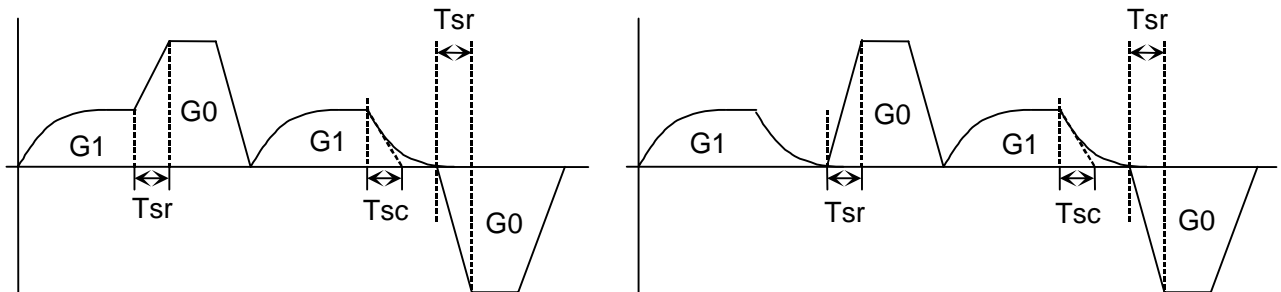
4.4 Acceleration / Deceleration

Acceleration / Deceleration during Continuing Blocks

(1) Continuous G1 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/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

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

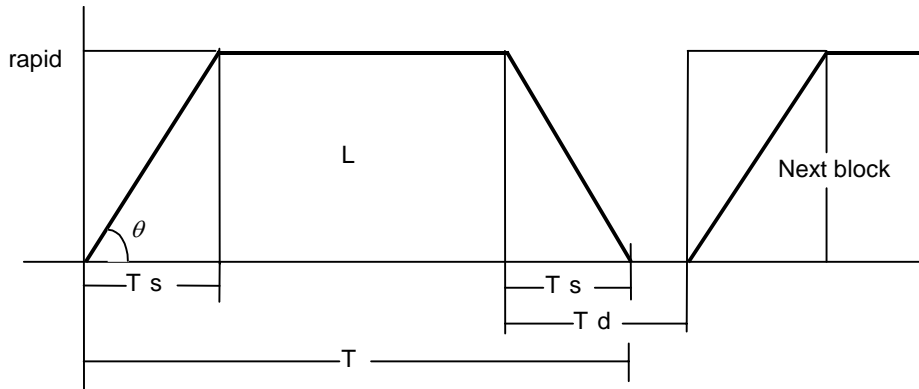
Rapid traverse constant inclination acceleration/deceleration are valid only for a rapid traverse command. Also, this function is effective only when the rapid traverse command acceleration/deceleration mode is linear acceleration and linear deceleration.

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

4 Feed

4.4 Acceleration / Deceleration

(1) When the interpolation distance is longer than the acceleration and deceleration distance



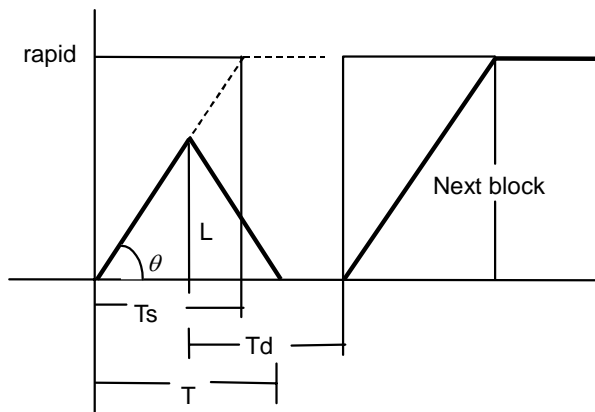
rapid : Rapid traverse rate
 Ts : Acceleration/deceleration time constant
 Td : Command deceleration check time
 θ : Acceleration/deceleration inclination
 T : Interpolation time
 L : Interpolation distance

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

$$T_d = T_s + (0 \sim 1.7 \text{ ms})$$

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

(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

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

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

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

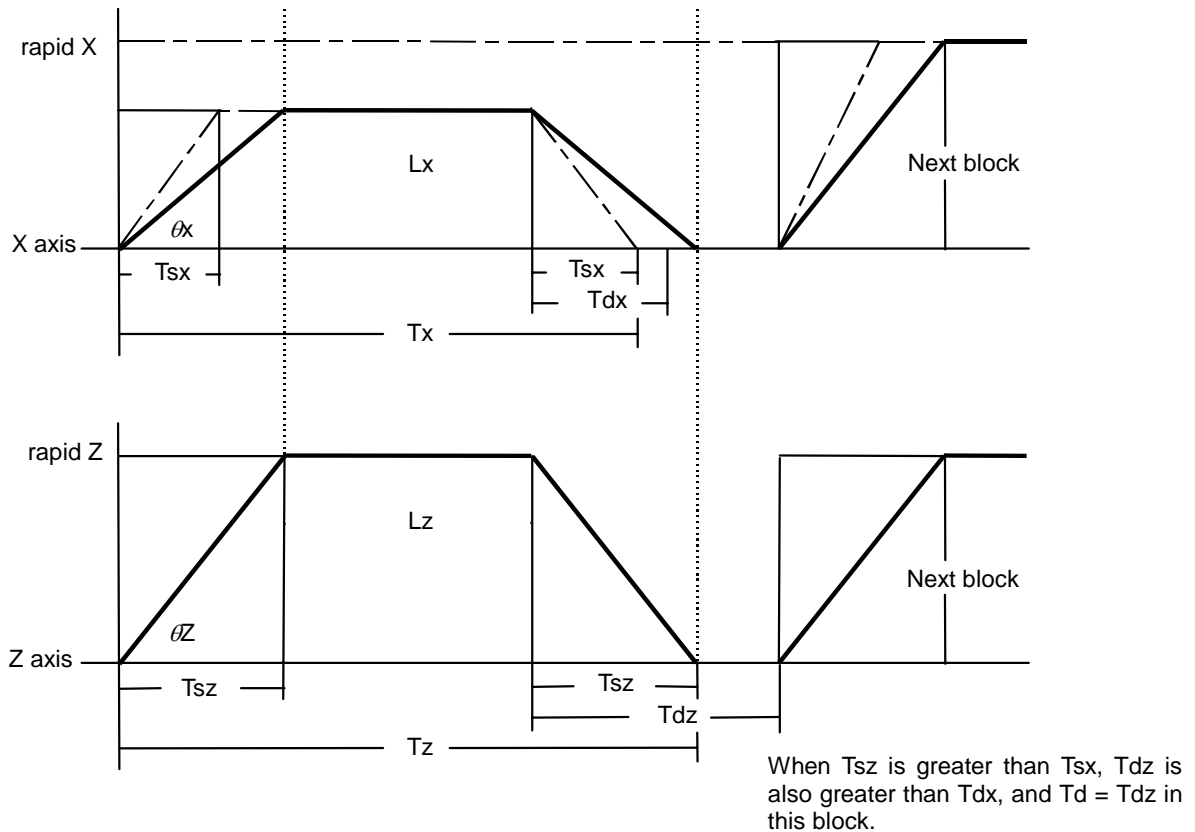
The time required to perform a command deceleration check during rapid traverse constant inclination acceleration/deceleration is the longest value among the rapid traverse deceleration check times determined for each axis by the rapid traverse rate of commands executed simultaneously, the rapid traverse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

4 Feed

4.4 Acceleration / Deceleration

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

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



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 Feed

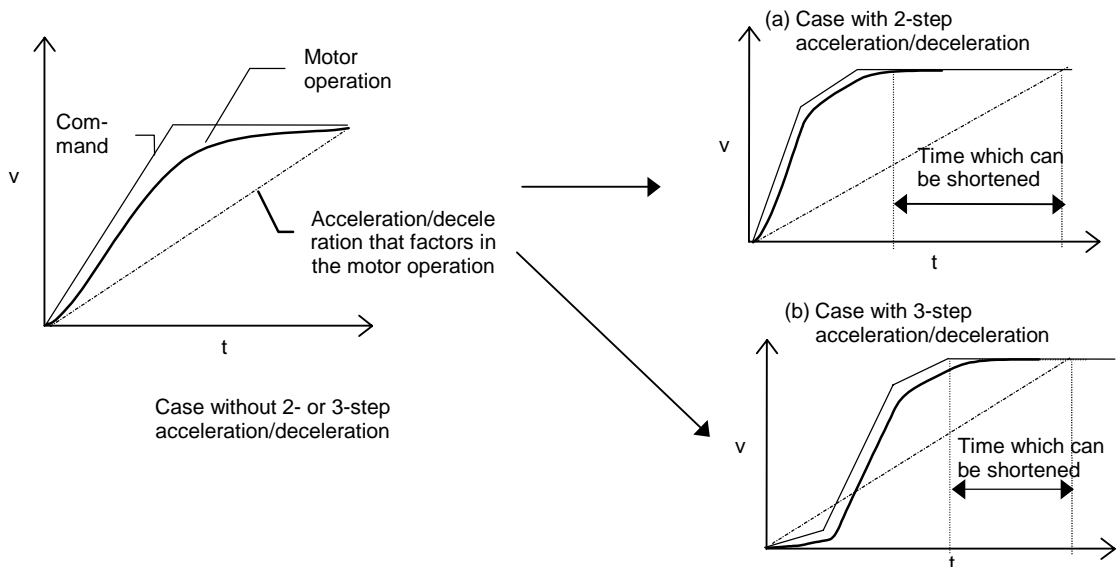
4.4 Acceleration / Deceleration

4.4.3 Rapid Traverse Constant Inclination 3-Step Acceleration / Deceleration

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	—	—	—	—	—	—
L system	○	○	—	—	—	—	—	—

The acceleration is normally calculated on the basis of speed/time when acceleration or deceleration is to be performed. In actual fact, however, the time taken for the desired positioning may increase due to changes (such as a reduction in the acceleration capability due to reduced torque at high-speed rotation) in the acceleration caused by the characteristics of the motor.

To offset this, the acceleration in the high rotation speed range is set separately from what is set for the low rotation speed range for a multi-step acceleration configuration. By so doing, the speed is controlled to match the characteristics of the motor so that acceleration can be performed up to the high rotation speed range while holding down the increase in the positioning time. This function is valid only during rapid traverse inclined constant linear acceleration/deceleration and G0 interpolation.



4 Feed

4.5 Thread Cutting

4.5 Thread Cutting

4.5.1 Thread Cutting (Lead/Thread Number Designation)

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	△	△	△	△	△	△	△	△
L system	○	○	○	○	○	○	○	○

(1) Lead designation

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

G33	Zz1	Qq1	Ff1/Ee1	;
G33	: Thread command			
Zz1	: Thread length			
Qq1	: Shift angle ("q1" is the shift angle at thread cutting start 0 to 360°)			
Ff1	: Thread lead			
Ee1	: Thread lead (precise lead threads)			

The tables below indicate the thread lead ranges.

[M system]

Metric command			Inch command		
Command increment (mm)	F (mm/rev)	E (mm/rev)	Command increment (inch)	F (inch/rev)	E (inch/rev)
0.001	0.001 ~999.999	0.00001 ~999.99999	0.0001	0.0001 ~39.3700	0.000001 ~39.370078
0.0001	0.0001 ~99.9999	0.000001 ~99.999999	0.00001	0.00001 ~3.93700	0.000001 ~3.937007

[L system]

Metric command			Inch command		
Command increment (mm)	F (mm/rev)	E (mm/rev)	Command increment (inch)	F (inch/rev)	E (inch/rev)
0.001	0.0001 ~999.9999	0.00001 ~999.99999	0.0001	0.00001 ~99.999999	0.000010 ~9.9999999
0.0001	0.00001 ~99.99999	0.000001 ~99.99999	0.00001	0.000001 ~9.9999999	0.0000010 ~0.99999999

The axis direction characterized by a large movement serves as the reference for the lead.

4 Feed

4.5 Thread Cutting

(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 Qq1 Ee1 ;
G33 : Thread command
Zz1 : Thread length
Qq1 : Shift angle ("q1" is the shift angle at thread cutting start 0 to 360°)
Ee1 : Thread number per inch

The tables below indicate the thread leads.

[M system]

Metric command		Inch command	
Command increment (mm)	Thread number command range (thread/inch)	Command increment (inch)	Thread number command range (thread/inch)
0.001	0.03~999.99	0.0001	0.0255~9999.9999
0.0001	255~9999.999	0.00001	0.25401~999.9999

[L system]

Metric command		Inch command	
Command increment (mm)	Thread number command range (thread/inch)	Command increment (inch)	Thread number command range (thread/inch)
0.001	0.03~999.99	0.0001	0.0101~9999.9999
0.0001	0.255~9999.999	0.00001	0.10001~999.99999

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

4 Feed

4.5 Thread Cutting

4.5.2 Variable Lead Thread Cutting

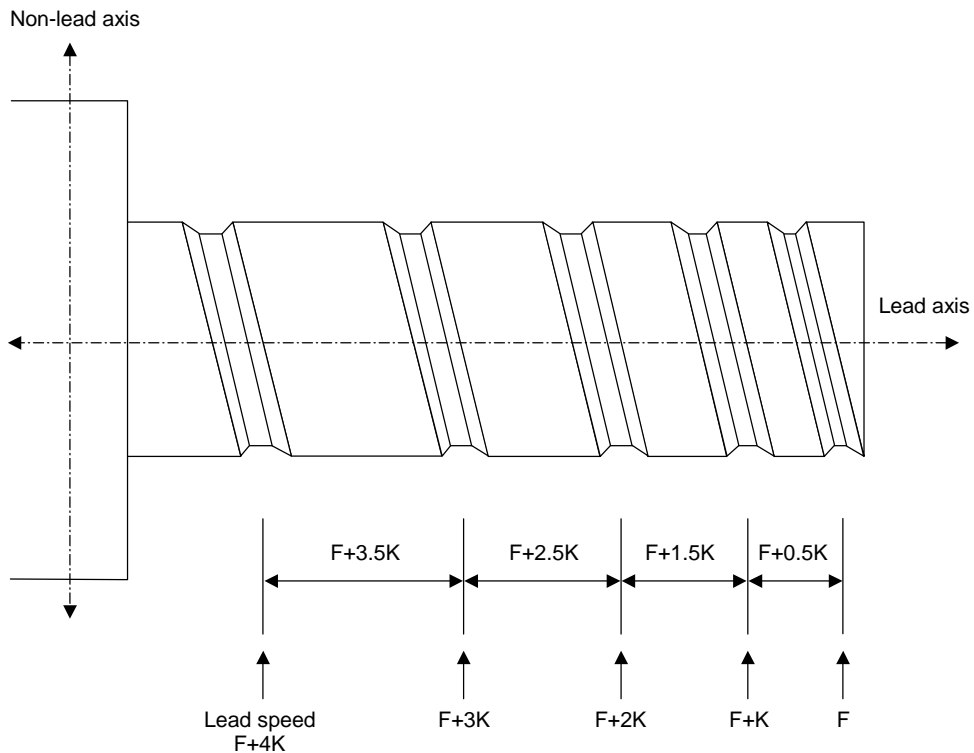
M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	-	-	-
L system	○	○	○	○	○	○	○	○

By commanding the lead increment/decrement amount per thread rotation, variable lead thread cutting can be done.

The machining program is commanded in the following manner.

G34 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 Feed

4.5 Thread Cutting

4.5.3 Synchronous Tapping

4.5.3.1 Synchronous Tapping Cycle

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ

This function performs tapping through the synchronized control of the spindle and servo. 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	: Mode, forward tapping
G74	: Mode, 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	: Mode, forward tapping
G74	: Mode, 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

4 Feed

4.5 Thread Cutting

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

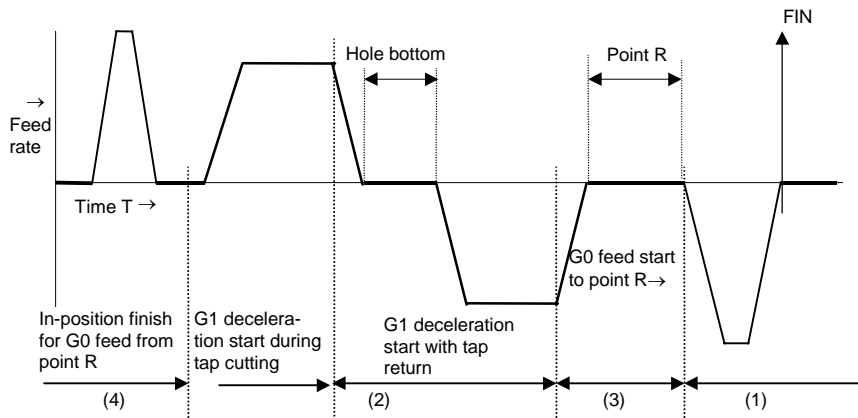
1. Cutting override Fixed to 100%
2. Feed hold invalid
3. "In tapping mode" signal is output
4. Deceleration command between blocks invalid
5. Single block invalid

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

(Note) The synchronous tapping cycle can be used for axes other than the Z axis with the plane selection.

Furthermore, in-position checks can be performed at the hole bottom or point R, etc. using the parameters. The figure below shows the correlation between the in-position width and the movement of the tapping axis of the synchronous tapping in-position check.



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

4 Feed

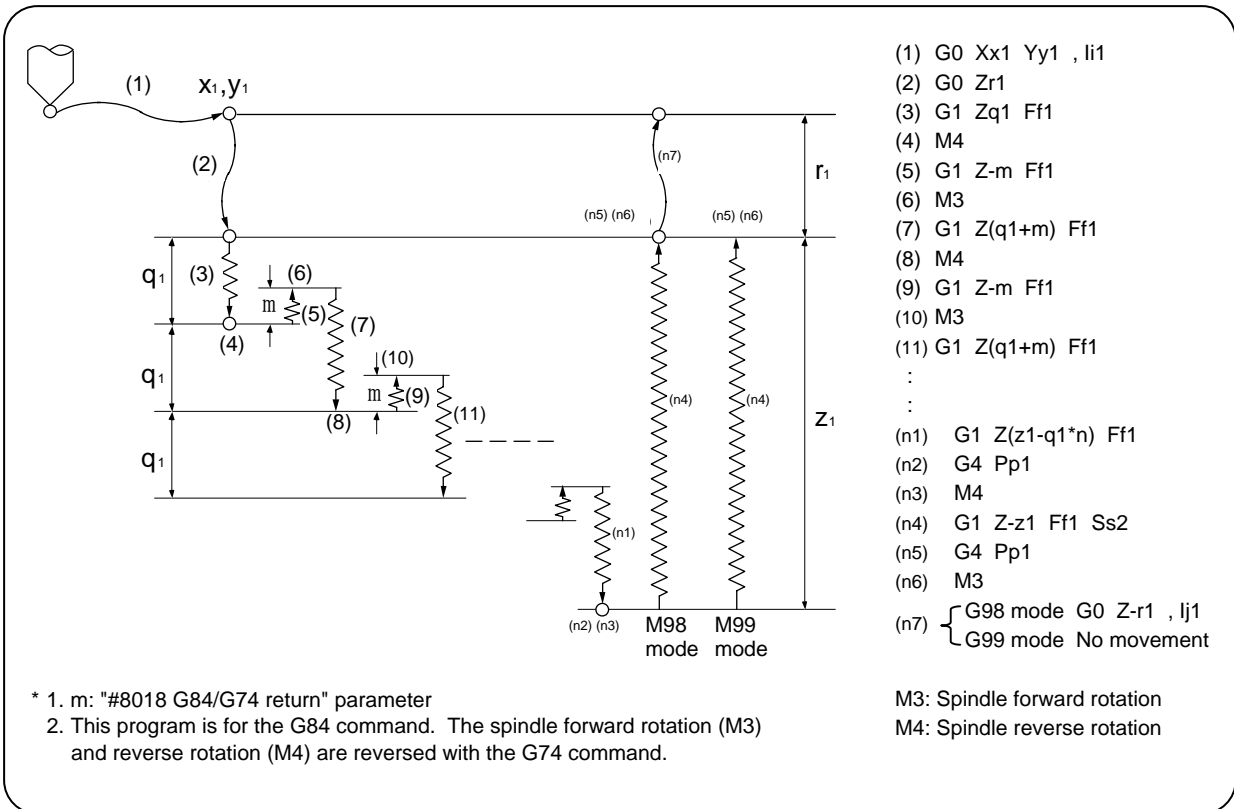
4.5 Thread Cutting

4.5.3.2 Pecking Tapping Cycle

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	-	-	-	-	-	-	-	-

G84(G74) Xx1 Yy1 Zz1 Rr1 Qq1 Ff1 Pp1 Ss1 ,Ss2 ,li1 ,Jj1 ,Rr2 ;

G : G84 forward tapping cycle, G74 reverse tapping cycle
X, Y : Hole drilling point position
Z : Hole bottom position
R : Point R position
Q : Depth of cut per pass (designated as an incremental value)
F : Cutting feed rate
P : Dwell time at hole bottom position
S : Rotation speed of spindle
,S : Rotation speed of spindle during retract
,I : In-position width of positioning axis
,J : In-position width of hole drilling axis
,R : r2=1 synchronous tapping mode, r2=0 asynchronous tapping mode



Adding this option, the deep-hole tapping cycle option is also added.
Select either the pecking tapping cycle or the deep-hole tapping cycle with parameter.
When the pecking tapping cycle is executed in the synchronous tapping mode, the synchronous tapping cycle option and pecking tapping cycle option are required.

4 Feed

4.5 Thread Cutting

4.5.3.3 Deep-hole Tapping Cycle

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	Δ	Δ	Δ	Δ
L system	-	-	-	-	-	-	-	-

In the deep-hole tapping, the load applied to the tool can be reduced by designating the depth of cut per pass and cutting the workpiece to the hole bottom for a multiple number of passes.

Adding this option, the pecking tapping cycle option is also added.

Select either the pecking tapping cycle or the deep-hole tapping cycle by parameter.

When the deep-hole tapping cycle is executed in the synchronous tapping mode, the synchronous tapping cycle option and deep-hole tapping cycle option are required.

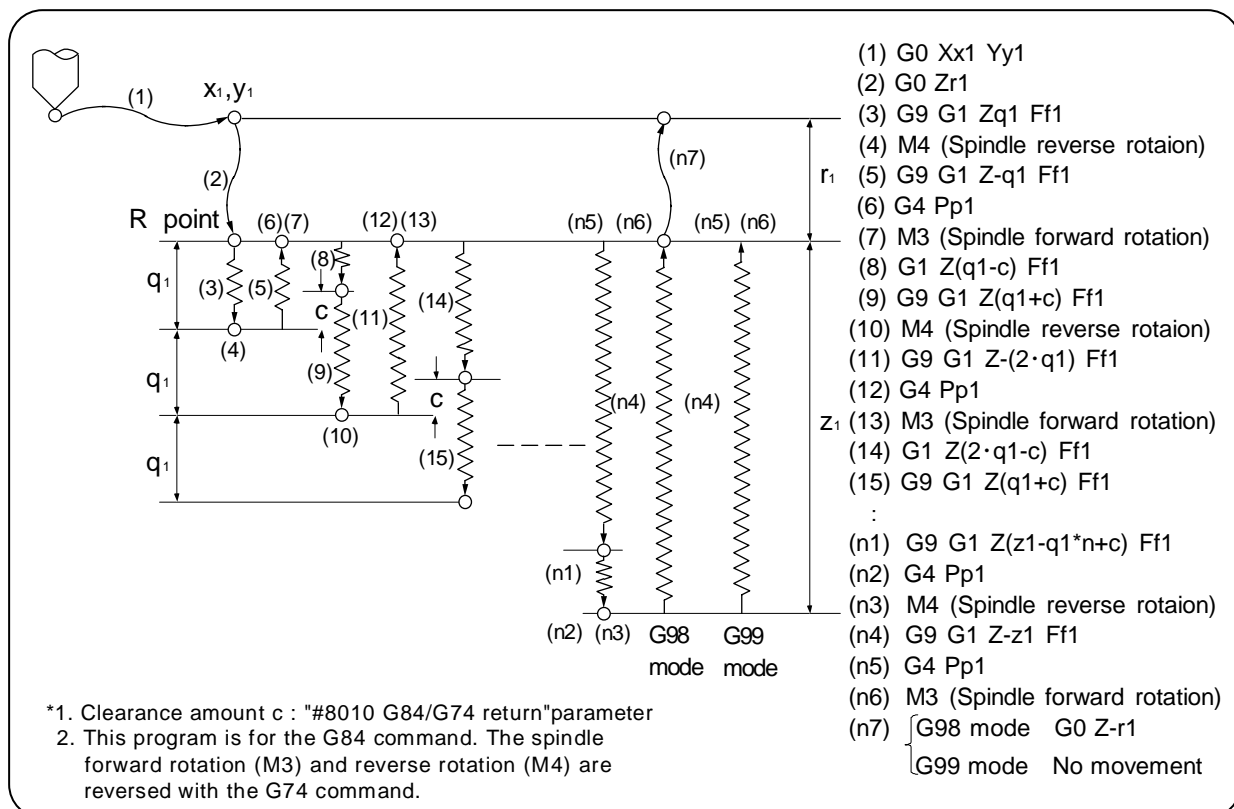
When "depth of cut per pass Q" is designated in the block containing the G84 or G74 tapping cycle command in the state when the deep-hole tapping cycle is selected by parameter, the deep-hole tapping cycle is executed.

In the following cases, the normal tapping cycle is established.

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

G84(G74) Xx1 Yy1 Zz1 Rr1 Qq1 Ff1 Pp1 ,Rr2 ;

G : G84 forward tapping cycle, G74 reverse tapping cycle
 X,Y : Hole drilling point position
 Z : Hole bottom position
 R : Point R position
 Q : Depth of cut per pass (designated as an incremental value)
 F : Cutting feed rate
 P : Dwell time at hole bottom and point R return
 ,R : r2=1 synchronous tapping mode, r2=0 asynchronous tapping mode



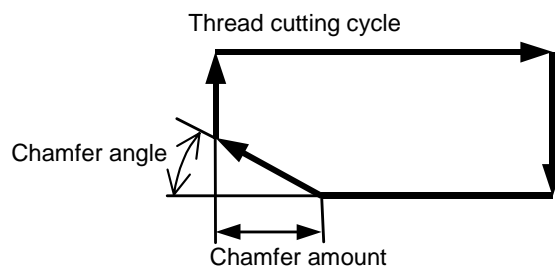
4 Feed

4.5 Thread Cutting

4.5.4 Chamfering

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	-	-	-
L system	○	○	○	○	○	○	○	○

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



4 Feed

4.5 Thread Cutting

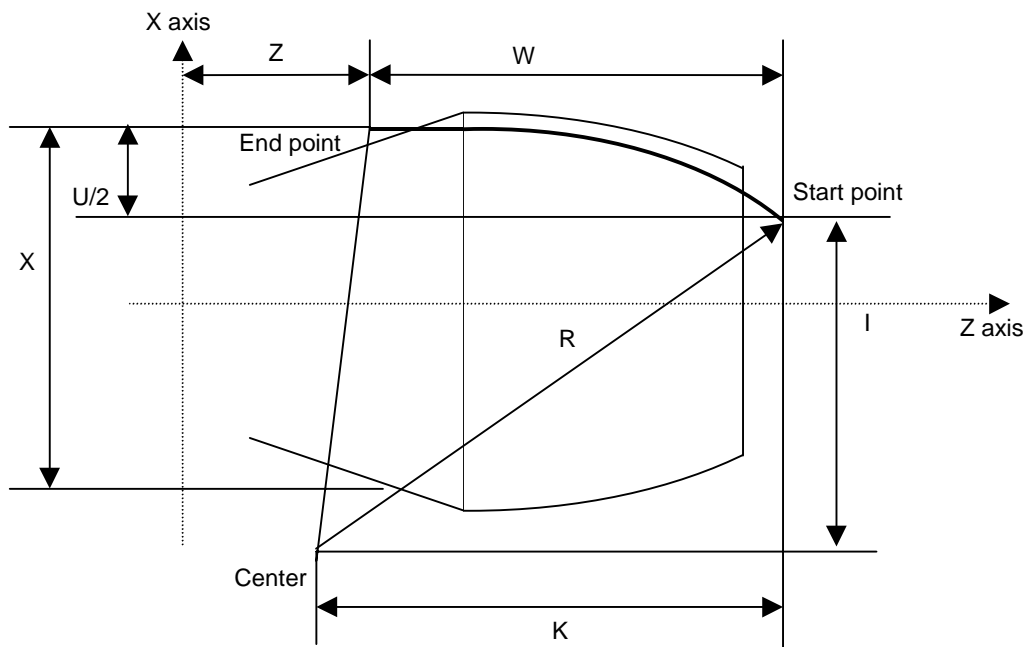
4.5.6 Circular Thread Cutting

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	-	-	-
L system	-	-	-	-	-	-	Δ	Δ

Circular threads in which the leads are in longitudinal direction can be cut.
Command format

G35 (G36) Xx/Uu Zz/Ww $\left\{ \begin{matrix} li & Kk \\ & Rr \end{matrix} \right\}$ **Ff/Ee Qq**

G35 : Clockwise (CW)
G36 : Counterclockwise (CCW)
Xx/Uu : X-axis arc end point coordinate (X: absolute value of workpiece coordinate system, U: incremental value from present position)
Zz/Ww : Z-axis arc end point coordinate (Z: absolute value of workpiece coordinate system, W: incremental value from present position)
li : X-axis arc center (incremental value of arc center as referenced from start point)
Kk : Z-axis arc center (incremental value of arc center as referenced from start point)
Rr : Arc radius
Ff/Ee : Longitudinal axis (axis with most travel) direction lead
(Ff: normal lead thread cutting, Ee: precision lead threads or inch threads)
Qq : Thread cutting start shift angle



4 Feed

4.6 Manual Feed

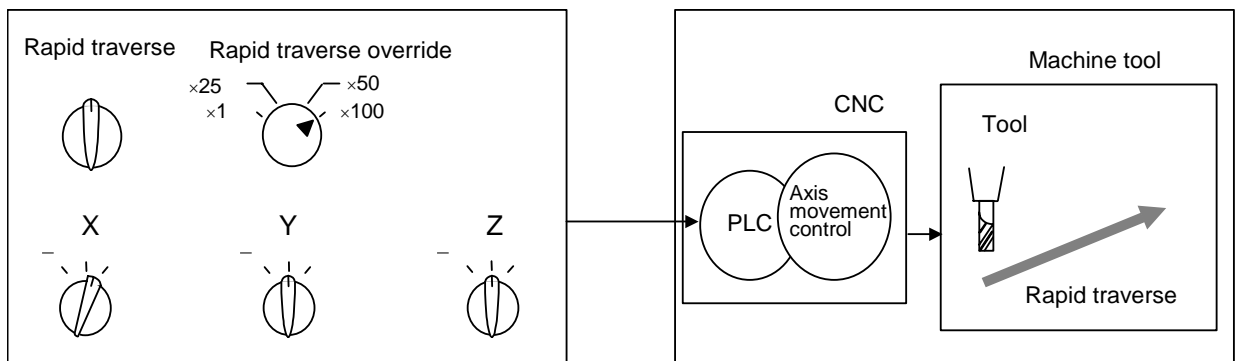
4.6 Manual Feed

4.6.1 Manual Rapid Traverse

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

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.

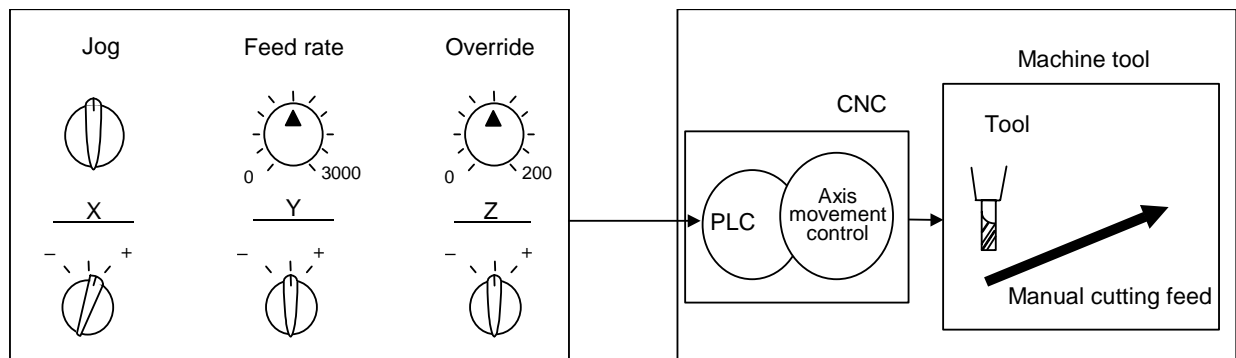
Rapid traverse override is common to all part systems.



4.6.2 Jog Feed

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

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



4 Feed

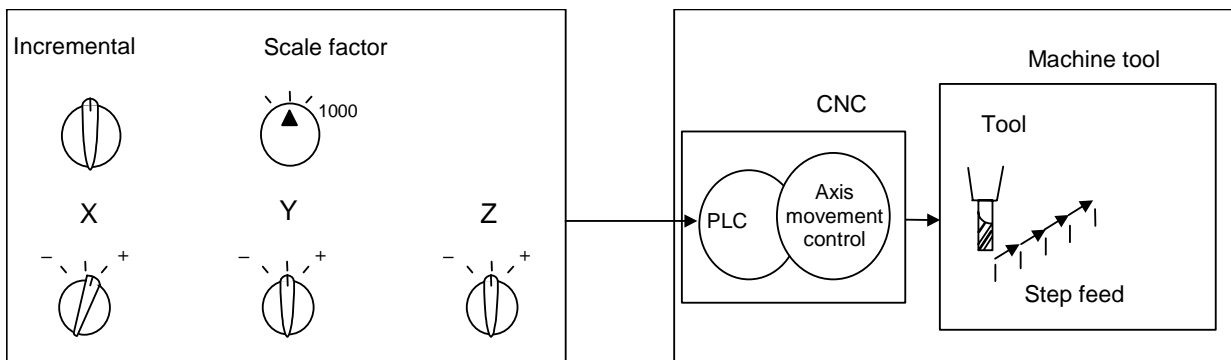
4.6 Manual Feed

4.6.3 Incremental Feed

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

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/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	△	△	△	△	△	△	△	△
L system	△	△	△	△	△	△	△	△

(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 random.

(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 Feed

4.6 Manual Feed

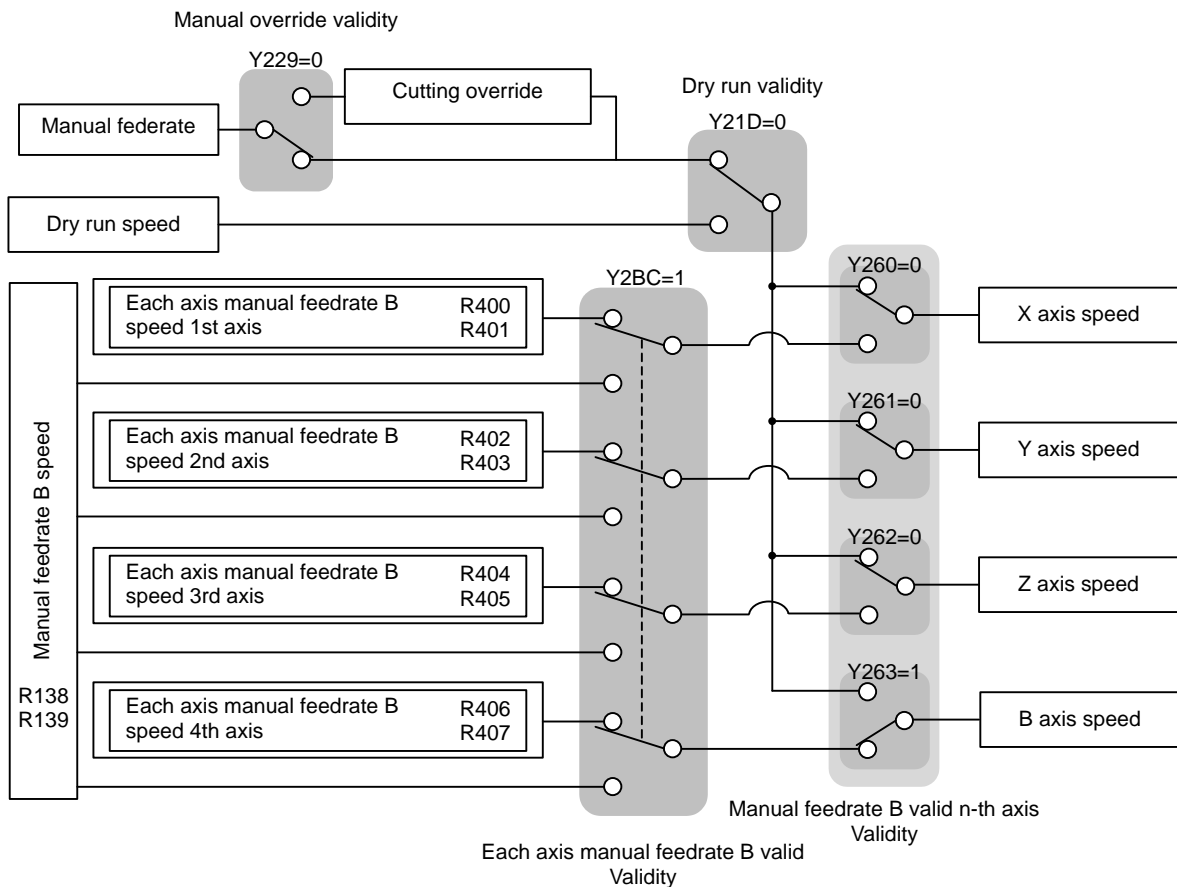
4.6.5 Manual Feed Rate B

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	△	△	△	△	○	○	○	○
L system	△	△	△	△	○	○	○	○

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

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

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



4 Feed

4.7 Dwell

4.7 Dwell

4.7.1 Dwell (Time-based Designation)

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

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

(G94) G04 Xx1/Uu1 ; or (G94) G04 Pp1 ;

G94 : Asynchronous

G04 : Dwell

Xx1, Uu1, Pp1 : Time

"x1" of the time-based dwell can be designated in the range from 0.001 to 99999.999 seconds.

5 Program Memory / Editing

5.1 Memory Capacity

5 Program Memory / Editing

5.1 Memory Capacity

Machining programs are stored in the NC memory.

5.1.1 Memory Capacity (Number of Programs Stored)

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

40 m (64 programs)

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		○	○	○	○	○	○	○	○
M system		○	○	○	○	○	○	○	○
L system		○	○	○	○	○	○	○	○

80 m (128 programs)

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		△	△	△	△	△	△	△	△
M system		△	△	△	△	△	△	△	△
L system		△	△	△	△	△	△	△	△

160 m (200 programs)

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		△	△	△	△	△	△	△	△
M system		△	△	△	△	△	△	△	△
L system		△	△	△	△	△	△	△	△

320 m (200 programs)

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		△	△	△	△	△	△	△	△
M system		△	△	△	△	△	△	△	△
L system		△	△	△	△	△	△	△	△

600 m (400 programs)

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		△	△	△	△	△	△	△	△
M system		△	△	△	△	△	△	△	△
L system		△	△	△	△	△	△	△	△

1280 m (1000 programs)

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		-	-	-	-	-	-	△	△
M system		-	-	-	-	-	-	△	△
L system		-	-	-	-	-	-	△	△

2560 m (1000 programs)

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		-	-	-	-	-	-	△	△
M system		-	-	-	-	-	-	△	△
L system		-	-	-	-	-	-	△	△

5120 m (1000 programs) M65S, M66S: When APLC is added, up to 2560m.

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		-	-	-	-	-	-	△	△
M system		-	-	-	-	-	-	△	△
L system		-	-	-	-	-	-	△	△

5 Program Memory / Editing

5.2 Editing

5.2 Editing Method

5.2.1 Program Editing

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

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.

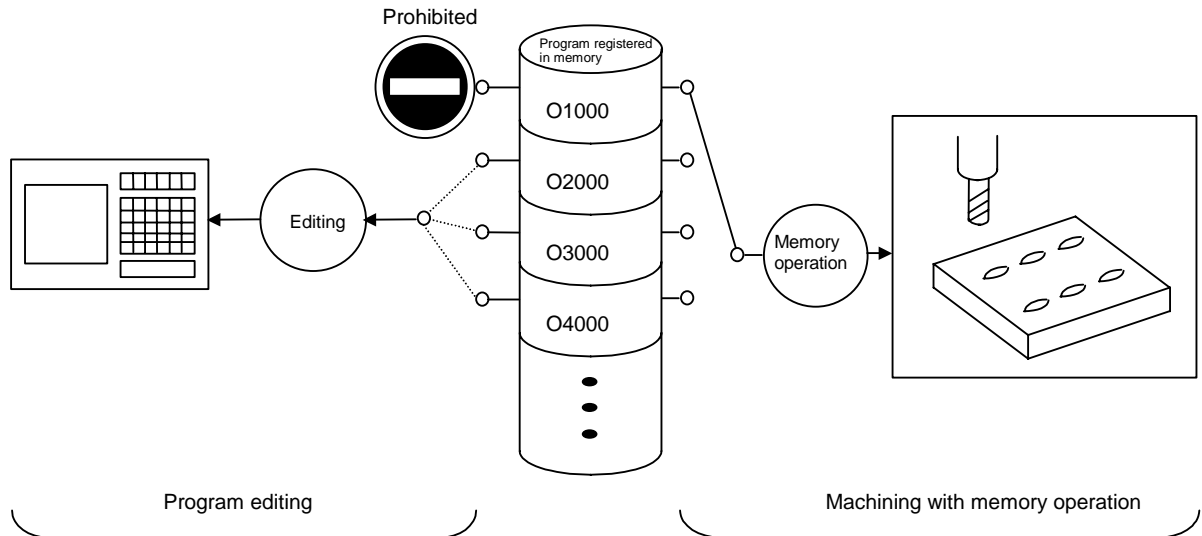
5 Program Memory / Editing

5.2 Editing

5.2.2 Background Editing

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

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

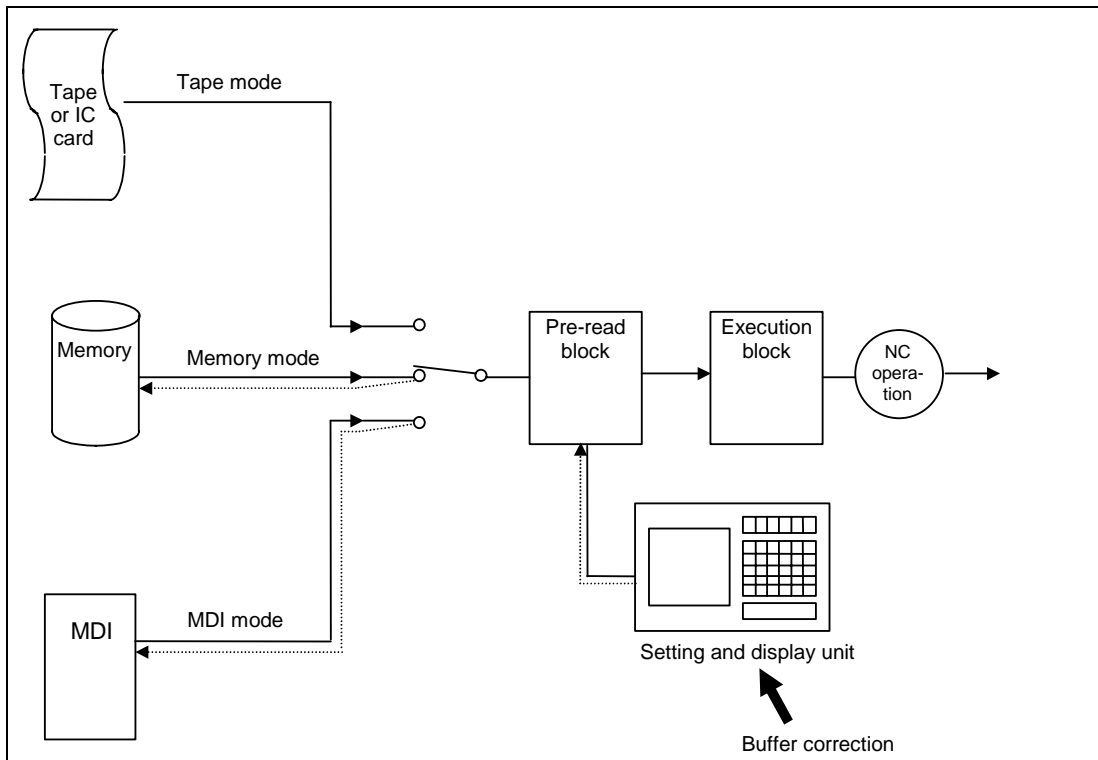
5 Program Memory / Editing

5.2 Editing

5.2.3 Buffer Correction

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

During automatic operation (memory or tape operation) or MDI operation, this function initiates single block stop and enables the next command to be corrected or changed.
 When a program error has occurred, the function enables the block in which the error occurred to be corrected and operation to be resumed without having to perform NC resetting.



5 Program Memory / Editing

5.2 Editing

5.2.4 Word Editing

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

In addition to the conventional editing function, this function enables programs to be edited in word increments.

It is also possible to create programs by deleting, replacing and inserting in word increments.

012345678 TEST CUT PROGRAM

EDIT

BACKGROUND EDITING

<SEARCH DATA>

<EDIT BUFFER>

>

SEARCH DELETE REPLACE INSERT MENU

```

N1 G28 X0 Y0 Z0 ;
N2 G00 X100.0 ;
N3 Z100.0 ;
N4 G01 X200.0 Z200.0 F500 ;
N5 X300.0 ;
N6 Z300.0 ;
N7 ;
N8 ;
N9 ;
N10 ;
N11 ;
N12 ;

```

<List of function>

Menu	Function
Delete	The word on which the cursor is positioned can be deleted. (A deleted word can also be un-deleted.)
Replace	The word on which the cursor is positioned can be replaced with editing buffer data. (The same word can also be repeatedly replaced.)
Insert	The editing buffer data can be inserted after the word on which the cursor is positioned. (The same word can also be repeatedly inserted.)
Copy	The word on which the cursor is positioned can be copied into the editing buffer. (The copied word can be used for replacement or insertion.)
Program	A list of the programs is displayed.
Operation search	Program numbers, sequence numbers and block numbers in the foreground can be searched.
Background search	Program numbers, sequence numbers and block numbers for background editing can be searched. New machining programs can be registered as well.
Background exit	Background editing is exited.
Comment	Comments can be set in machining programs.
Word ↓	A downward search for a word is conducted, and the cursor is moved to the word in question. (The same word can be repeatedly searched.)
Word ↑	An upward search for a word is conducted, and the cursor is moved to the word in question. (The same word can be repeatedly searched.)
String ↓	A downward search for a character string is conducted, and the cursor is moved to the words in question. (The same character string can be repeatedly searched.)
String ↑	An upward search for a character string is conducted, and the cursor is moved to the words in question. (The same character string can be repeatedly searched.)
Running program display	The program being run is displayed when the program running display request (PLC) is ON.
Program operation start position setting	The start block can be designated by moving the cursor on the editing screen.

6 Operation and Display

6.1 Structure of Operation / Display Panel

6 Operation and Display

6.1 Structure of Operation / Display Panel

The setting and display unit is configured of the setting section and keyboard section. When the key switches are pressed, a buzzer sounds allowing the operation to be confirmed visually and audibly. The following display units can be used for the setting and display unit.

Type	Function	Configuration element	Details
FCU6-DUT32	10.4-type monochrome LCD display unit (separated type)	10.4-type LCD, Escutcheon Control card (RX215)	Used as a set with FCUA-KB20. Control card 24VDC input
FCU6-DUN33	10.4-type color LCD display unit (separated type)	10.4-type LCD, Escutcheon Control card (RX215)	Used as a set with FCUA-KB20. Control card 24VDC input
FCUA-LD100	7.2-type monochrome LCD liquid crystal (integrated type)	7.2-type LCD, Escutcheon Control card (RX213) Key switches	Control card 24VDC input
FCUA-LD10	7.2-type monochrome LCD display unit (separated type)	7.2-type LCD, Escutcheon Control card (RX213)	Used as a set with FCUA-KB20. Control card 24VDC input
FCUA-CT100	9-type monochrome CRT + communication terminal (integrated type, machining center system sheet)	9-type monochrome CRT, Escutcheon Control card (RX211) Key switches	Control card 24VDC input CRT section 100VAC input
FCUA-CT120	9-type monochrome CRT + communication terminal (integrated type, lathe system sheet)	9-type monochrome CRT, Escutcheon Control card (RX211) Key switches	Control card 24VDC input CRT section 100VAC input
FCUA-CR10	9-type monochrome CRT communication terminal (separated type)	9-type monochrome CRT, Escutcheon	Used as a set with FCUA-KB10/KB12. Control card 24VDC input CRT section 100VAC input
FCUA-KB10	Communication terminal (separated type, machining center system sheet)	Key switch Control card	Used as a set with FCUA-CR10.
FCUA-KB12	Communication terminal (separated type, lathe system sheet)	Key switch Control card	Used as a set with FCUA-CR10.
FCUA-KB20	Communication terminal (separated type, machining center system sheet)	Key switch	Use as a set with FCU6-DUT32/ DUN33 or FCUA-LD10
FCUA-KB021	Communication terminal (separated type, machining center system sheet)	Key switch	Use as a set with FCU6-DUT32/ DUN33 KB20 with changed outline dimensions
FCUA-KB30	Communication terminal (separated type, lathe system sheet)	Key switch	Use as a set with FCU6-DUT32/ DUN33 or FCUA-LD10
FCU6-KB031	Communication terminal (separated type, lathe system sheet)	Key switch	Use as a set with FCU6-DUT32/ DUN33 KB30 with changed outline dimensions

6.2 Operation Methods and Functions

6.2.1 Memory Switch (PLC Switch)

M/L system	Model	M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system		○	○	○	○	○	○	○	○
L system		○	○	○	○	○	○	○	○

The toggle switches (PLC switches) can be defined on the screen.

The screen can be operated by turning the switches ON/OFF, and the status can be read from the PLC ladder. This screen has been prepared in advance, so the switch names (display on screen) can be defined with the PLC ladder.

There are a total of 32 switch points.

6 Operation and Display

6.3 Display Methods and Contents

6.3 Display Methods and Contents

6.3.1 Status Display

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

The status of the program now being executed is indicated.

- (1) Display of G, S, T, M commands and 2nd miscellaneous command modal values
- (2) Feed rate display
- (3) Tool offset number and offset amount display
- (4) Real speed display (*)

(*) The feed rate of each axis is converted from the final speed output to the drive amplifier, 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 Position Display

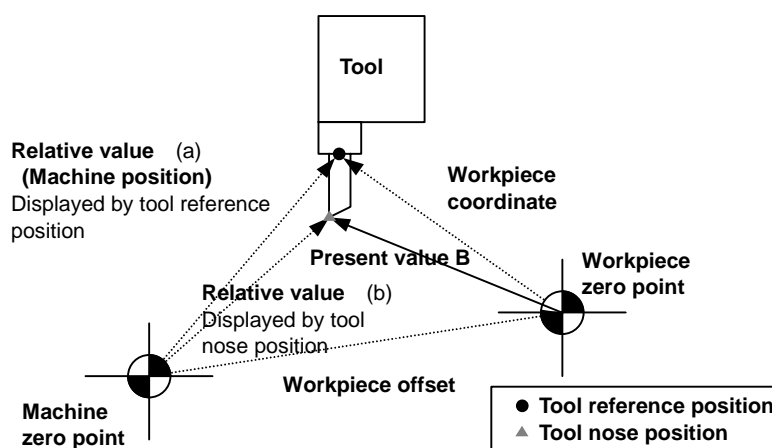
Position data such as present positions for tools, coordinate positions and workpiece coordinate positions can be displayed.

- (1) Present position counter

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

Each axis' present value including tool length offset amount, tool radius compensation amount and workpiece coordinate offset amount is indicated.

Whether the tool reference position (figure below (a)) or the present position of the tool nose position (figure below (b)) that considers offset, such as tool length offset amount or tool diameter compensation amount, in the tool reference position is applied to the display of the relative value can be selected with the parameter.



6 Operation and Display

6.3 Display Methods and Contents

(2) Workpiece coordinate counter

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
	M system	○	○	○	○	○	○	○	○
	L system	○	○	○	○	○	○	○	○

The workpiece coordinate system modal number from G54 to G59 and the workpiece coordinate value in the workpiece coordinate system are indicated.

(3) Remaining command counter

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
	M system	○	○	○	○	○	○	○	○
	L system	○	○	○	○	○	○	○	○

The remaining distance of the movement command during the execution (incremental distance from the present value to the end point of the block) is indicated during the automatic start and automatic stop.

(4) Machine position counter

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
	M system	○	○	○	○	○	○	○	○
	L system	○	○	○	○	○	○	○	○

Each axis' coordinate value in the basic machine coordinate system whose zero point is the characteristic position determined depending on the machine is indicated.

(5) Present value B

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
	M system	–	–	–	–	○	○	○	○
	L system	○	○	○	○	○	○	○	○

Each axis' value not including tool length offset amount, tool radius compensation amount and workpiece coordinate offset amount is indicated.

Whether the counter value on the Position screen is expressed with the Present or with the present value B can be selected using parameter.

The present value B can be selected for the counter value indicated on the coordinate value screen using parameter.

(6) Manual interrupt amount counter

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
	M system	–	–	–	–	○	○	○	○
	L system	–	–	–	–	○	○	○	○

The amount moved with the manual mode while the manual absolute switch was OFF is indicated. On the coordinate value screen, in addition to the manual interrupt amount, the MST display, next command counter and present value B can be selected for the indicated counter using parameter.

6 Operation and Display

6.3 Display Methods and Contents

6.3.3 Program Running Status display

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>
M system		<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>
L system		<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>

Program now being executed is displayed.

6.3.4 Setting and Display

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>
M system		<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>
L system		<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>

The parameters used in controller operations can be set and displayed.

6.3.5 MDI Data Setting and Display

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>
M system		<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>
L system		<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>

The MDI data having a multiple number of blocks can be set and displayed. As with the editing of machining programs, the MDI programs can be revised using the delete, change and add functions. Operation can be repeated using the programs which have been set.

6.3.6 Specification List Display

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		–	–	–	–	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>
M system		–	–	–	–	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>
L system		–	–	–	–	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>

This function displays a list of the specifications which are currently valid.

6.3.7 Clock

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>
M system		<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>
L system		<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>

The clock is built-in, and the date and time are displayed.
 Once the time is set, it can be seen as a clock on the screen.
 The clock time can be read/written (read/set) from PLC using the DDB function.

6 Operation and Display

6.3 Display Methods and Contents

6.3.8 Hardware / Software Configuration Display

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		○	○	○	○	○	○	○	○
M system		○	○	○	○	○	○	○	○
L system		○	○	○	○	○	○	○	○

This function displays the configuration of the installed hardware and software.

6.3.9 Integrated Time Display

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		○	○	○	○	○	○	○	○
M system		○	○	○	○	○	○	○	○
L system		○	○	○	○	○	○	○	○

The integrating run time count during each signal of power-ON, automatic operation, automatic start and external integrating run time is ON can be set and displayed. The maximum time displayed is 9999 hours 59 minutes 59 seconds.

Power-ON: Total of all the integrated run times, each starting when the power of the NC control unit is turned ON and ending when it is turned OFF.

Automatic operation: Total of the integrated run times for all machining periods, each starting when the auto start button is pressed in the memory mode and ending when the reset status is established (usually when the M02 / M30 command is designated or the reset button is pressed). (This differs according to PLC machining.)

Automatic start: Total of the integrated run times for all automatic start operations, each starting when the auto start button is pressed in the memory or MDI mode and ending when the feed hold stop or block stop is established or the reset button is pressed.

External integration: Based on the PLC sequence, this is the integrated run time of the signal set by the PLC, and it comes in two types, external integration 1 and external integration 2.

6.3.10 Available Languages

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		○	○	○	○	○	○	○	○
M system		12 languages	12 languages	12 languages	12 languages	13 languages	13 languages	13 languages	13 languages
L system		○	○	○	○	○	○	○	○
		12 languages	12 languages	12 languages	12 languages	13 languages	13 languages	13 languages	13 languages

This function makes it possible to switch between Japanese and English which are the standard languages used for the screen displays.

Furthermore, under the optional specifications, the display language can be switched to German, Italian, French, Spanish, Chinese (Traditional Chinese characters, simplified Chinese characters), Korean, Portuguese, Hungarian, Dutch or Swedish.

6 Operation and Display

6.3 Display Methods and Contents

6.3.11 Additional Languages

(Japanese, English, German, Italian, French, Spanish, Chinese, Korean, Portuguese, Hungarian, Dutch, Swedish)

6.3.11.1 Japanese

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		○	○	○	○	○	○	○	○
M system		○	○	○	○	○	○	○	○
L system		○	○	○	○	○	○	○	○

6.3.11.2 English

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		○	○	○	○	○	○	○	○
M system		○	○	○	○	○	○	○	○
L system		○	○	○	○	○	○	○	○

6.3.11.3 German

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		△	△	△	△	△	△	△	△
M system		△	△	△	△	△	△	△	△
L system		△	△	△	△	△	△	△	△

6.3.11.4 Italian

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		△	△	△	△	△	△	△	△
M system		△	△	△	△	△	△	△	△
L system		△	△	△	△	△	△	△	△

6.3.11.5 French

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		△	△	△	△	△	△	△	△
M system		△	△	△	△	△	△	△	△
L system		△	△	△	△	△	△	△	△

6.3.11.6 Spanish

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		△	△	△	△	△	△	△	△
M system		△	△	△	△	△	△	△	△
L system		△	△	△	△	△	△	△	△

6 Operation and Display

6.3 Display Methods and Contents

6.3.11.7 Chinese

(1) Chinese (Traditional Chinese characters)

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	△	△	△	△	△	△	△	△
L system	△	△	△	△	△	△	△	△

(2) Chinese (Simplified Chinese characters)

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	△	△	△	△
L system	-	-	-	-	△	△	△	△

6.3.11.8 Korean

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	△	△	△	△	△	△	△	△
L system	△	△	△	△	△	△	△	△

6.3.11.9 Portuguese

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	△	△	△	△	△	△	△	△
L system	△	△	△	△	△	△	△	△

6.3.11.10 Hungarian

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	△	△	△	△	△	△	△	△
L system	△	△	△	△	△	△	△	△

6.3.11.11 Dutch

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	△	△	△	△	△	△	△	△
L system	△	△	△	△	△	△	△	△

6.3.11.12 Swedish

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	△	△	△	△	△	△	△	△
L system	△	△	△	△	△	△	△	△

6 Operation and Display

6.3 Display Methods and Contents

6.3.12 Screen Saver, Backlight OFF

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	○	○	○	○
L system	-	-	-	-	○	○	○	○

The screen saver and backlight OFF functions turn off the displays when there is no need to view the screen.

This function is available only on the LCD.

<Screen saver>

This function protects the screen display unit by blanking the screen after the time set in the parameter has elapsed.

Inputting any key causes the screen to reappear.

<Backlight OFF>

This function turns off the backlight in order to extend the service life of the LCD screen's backlight.

6.3.13 Screen Deletion

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

When there is no need to use a screen for extended periods, the entire screen can be cleared to prevent deterioration of the display unit by the following procedures.

7 Input / Output Functions and Devices

7.1 Input / Output Data

7 Input / Output Functions and Devices

7.1 Input / Output Data

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

Machining program input / output (including user macros and fixed cycle macros)

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
	M system	○	○	○	○	○	○	○	○
	L system	○	○	○	○	○	○	○	○

Tool offset data input / output

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
	M system	○	○	○	○	○	○	○	○
	L system	○	○	○	○	○	○	○	○

Common variable input / output

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
	M system	○	○	○	○	○	○	○	○
	L system	○	○	○	○	○	○	○	○

Parameter input / output

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
	M system	○	○	○	○	○	○	○	○
	L system	○	○	○	○	○	○	○	○

History data output

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
	M system	○	○	○	○	○	○	○	○
	L system	○	○	○	○	○	○	○	○

Remote program input

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
	M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
	L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ

7 Input / Output Functions and Devices

7.2 Input / Output I/F

7.2 Input / Output I/F

7.2.1 RS-232C I/F

Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

Port 1 and 2 of the RS-232C interface can be used.

Port	Port 1/2
Transmission speed	~ 19.2kbps
Handshake method	DC code method, RTS/CTS method possible

Each port can be used for the following application.

Port 1: Input/output, tape operation, Anshin-net

Port 2: Input/output, tape operation, GX Developer communication, computer link, Anshin-net

7.2.2 IC Card I/F

7.2.2.1 I/F for IC card in Control Unit

Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	Δ	Δ	Δ	Δ	Δ	Δ
L system	-	-	-	-	Δ	Δ	Δ	Δ

IC cards can be attached inside the NC unit and used.

7.2.3 Data Server I/F

Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	Δ	Δ	-	Δ	Δ	Δ
L system	-	-	-	-	-	-	-	-

Ethernet I/F cards can be attached onto the NC unit and used.

(The Ethernet for the data server interface is the same as that for the Ethernet interface described in section 7.2.4.)

7.2.4 Ethernet I/F

Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	Δ	Δ	Δ	Δ	Δ	Δ
L system	-	-	-	-	Δ	Δ	Δ	Δ

Ethernet I/F card can be attached onto the NC unit and used.

7 Input / Output Functions and Devices

7.3 Computer Link

7.3 Computer Link

7.3.1 Computer Link B

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ

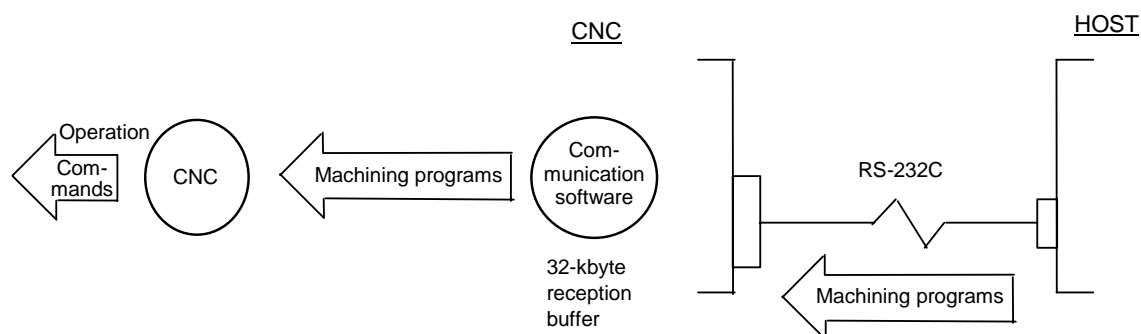
This function sends DC1 to the host computer (hereafter abbreviated to "HOST") at the CNC cycle start, and it enables operation to be performed while the machining programs are received from the host.

The computer link has a 32-kbyte reception buffer so that operation will be less susceptible to the effects of the data transfer status at the HOST end. This means that when the machine is connected to a HOST capable of transferring data at a high speed (of 38,400 bps), it is possible to perform

high-speed machining based on fine-segment data.

The high-speed machining mode option is required for high-speed fine-segment machining.

When using the 2-part system, this function cannot be operated in the 2nd part system.



BTR operation

8 Spindle, Tool and Miscellaneous Functions

8.1 Spindle Functions (S)

8 Spindle, Tool and Miscellaneous Functions

8.1 Spindle Functions (S)

8.1.1 Command / Output

8.1.1.1 Spindle Functions

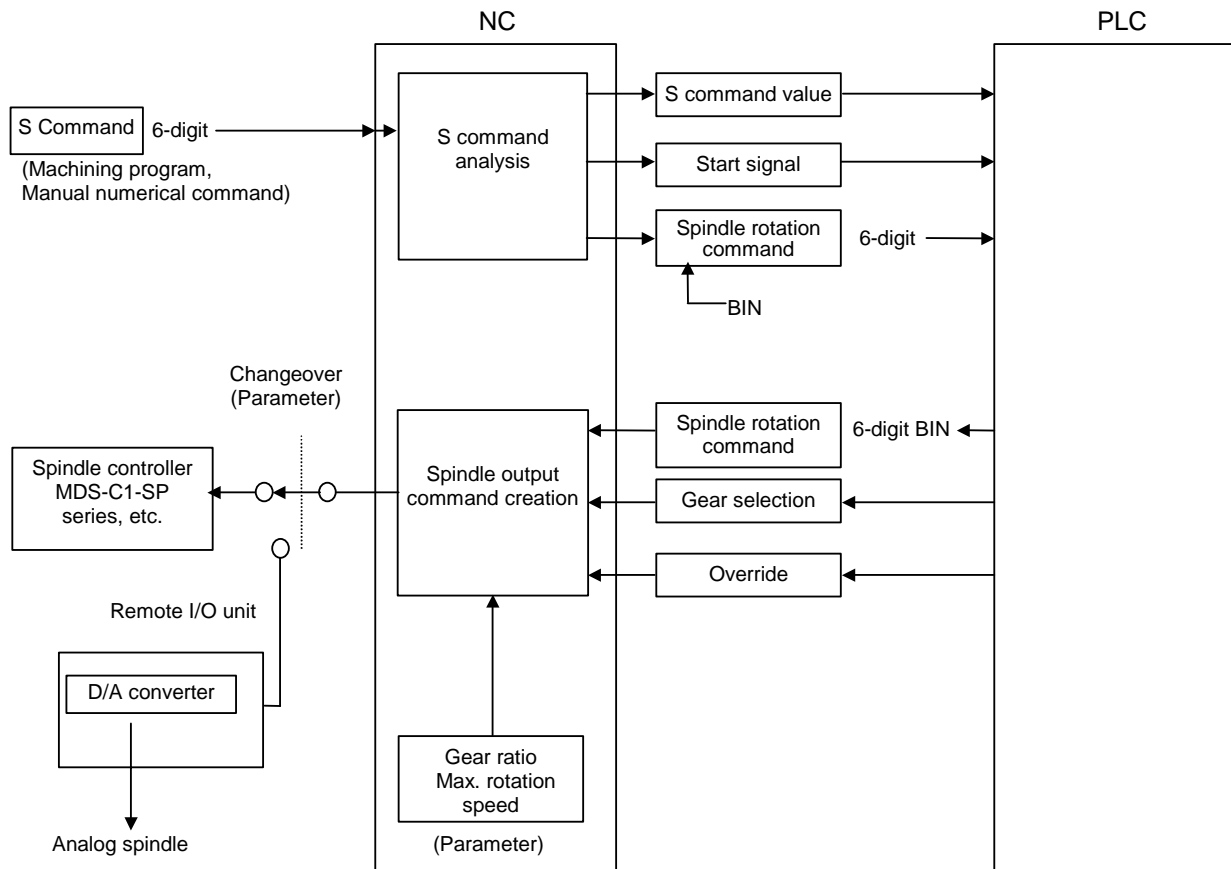
M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

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

When an 8-digit number following address S (S00000000 to S±99999999) is commanded, a signed 32-bit binary data or 8-digit BCD data and start signal will be output to the PLC.

Only one set of S commands can be commanded in one block.

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



8 Spindle, Tool and Miscellaneous Functions

8.1 Spindle Functions (S)

- (1) The override can be designated as 50% to 120% in 10% increments or 0 to 200% in 1% increments (with built-in PLC specifications).

The override is not changed while the spindle stop input is ON, during the tapping mode, or during the thread cutting mode.

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

8.1.1.2 Spindle Serial I/F

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		○	○	○	○	○	○	○	○
M system		○	○	○	○	○	○	○	○
L system		○	○	○	○	○	○	○	○

This I/F is used to connect the digital spindle (AC spindle motor and spindle driver (SP, SPJ2)).

8.1.1.3 Spindle Analog I/F

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
M system		Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system		Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ

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

In this case, the remote I/O unit DX120/DX121 is required.

The analog output voltage is calculated from the present rotation speed regarding the voltage at the max. rotation speed as the maximum analog voltage.

The specifications of the analog voltage output are as follows.

- (1) Output voltage ... 0 to 10 V
 (2) Resolution ... 1/4095 (–12 multiplier of 2)
 (3) Load conditions ... 10 kΩ
 (4) Output impedance ... 220 Ω

8.1.1.4 Coil Change

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		○	○	○	○	○	○	○	○
M system		○	○	○	○	○	○	○	○
L system		○	○	○	○	○	○	○	○

Constant output characteristics can be achieved across a broad spectrum down to the low-speed range by switching the spindle motor connections.

This is a system under which commands are assigned from the PLC.

8.1.1.5 Automatic Coil Change

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		–	–	–	–	○	○	○	○
M system		–	–	–	–	○	○	○	○
L system		–	–	–	–	○	○	○	○

Constant output characteristics can be achieved across a broad spectrum down to the low-speed range by switching the spindle motor connections.

This is a system under which the NC unit switches the coils automatically in accordance with the motor speed.

8 Spindle, Tool and Miscellaneous Functions

8.1 Spindle Functions (S)

8.1.2 Speed Control

8.1.2.1 Constant Surface Speed Control

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	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.

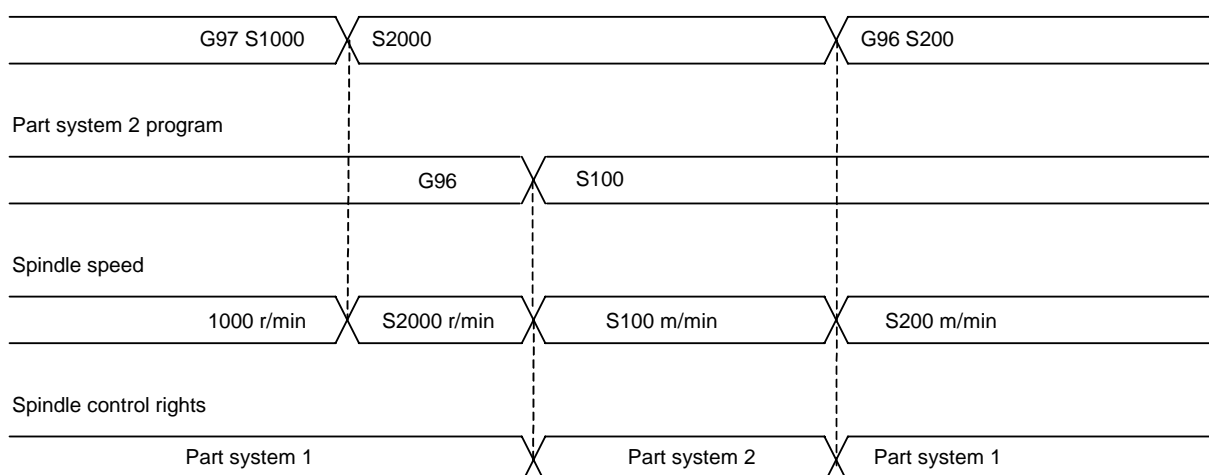
In the constant surface speed cancel mode, the S code is a spindle rotation speed command.

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) 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.

Part system 1 program



8 Spindle, Tool and Miscellaneous Functions

8.1 Spindle Functions (S)

8.1.2.2 Spindle Override

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

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

(1) Type 1 (code method)

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

(2) Type 2 (number setting method)

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

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

8.1.2.3 Multiple-spindle Control

Multi-spindle control is a function that controls all the spindles except for the first spindle for a machine tool equipped with second, third and fourth spindles (sub-spindles) in addition to the first spindle (main spindle).

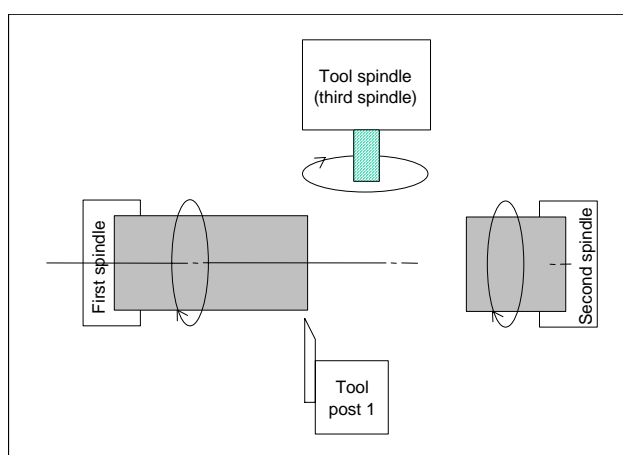
Multi-spindle control I can be switched to multi-spindle control II or vice versa using a parameter and, by so doing, the spindle control method changes.

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

Multi-spindle control II: Control based on an external signal (spindle command selection signal, spindle selection signal) and spindle control command ([S****;] only), etc.

Spindle selection commands [SO=****;] cannot be used for this control.

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



8 Spindle, Tool and Miscellaneous Functions

8.1 Spindle Functions (S)

8.1.2.3.1 Multiple-spindle Control I

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	–	Δ	Δ	Δ	–	Δ	Δ	Δ
L system	–	Δ	Δ	Δ	–	Δ	Δ	Δ

Spindle selection function

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 fourth spindles to which the subsequent S command (S*****) is to apply.

Command format

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

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 fourth spindles by using the SO=*****.

Command format

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

8.1.2.3.2 Multiple-spindle Control II

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	–	Δ	Δ	Δ	–	Δ	Δ	Δ
L system	–	Δ	Δ	Δ	–	Δ	Δ	Δ

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

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

Spindle command selection, spindle selection

The S command for the spindle is output as the rotation speed command to the spindle which has been selected by the spindle selection signal (SWS) ON from the PLC. The selected spindle rotates at the rotation speed which was output. The spindles which were de-selected by spindle selection signal (SWS) OFF continue to rotate at the same rotation speed as the speed immediately before their de-selection. As a result, each of the spindles can be made to rotate simultaneously at a different rotation speed. The spindle command selection signal is used to select which of the spindles is to receive the S command from which system.

8 Spindle, Tool and Miscellaneous Functions

8.1 Spindle Functions (S)

8.1.3 Position Control

8.1.3.1 Spindle Orientation

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

(a) Orient

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

When the orient command is used, the spindle will rotate several times and then stop at the orient point. The orient point is the Z-phase position when using encoder orient (PLG and external encoder/ring sensor).

(b) Multi-point orient

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 4095. (Unit: 360°/4096)

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

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

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

(c) Pre-positioning orient (spindle orient 2)

When the in-position pre-positioning parameter is valid and the second in-position is valid, this function turns ON the orient finish signal as soon as the spindle reaches within the pre-positioning in-position width. It also turns ON the second in-position signal as soon as the spindle reaches within the orient in-position width. (OINP) actually.

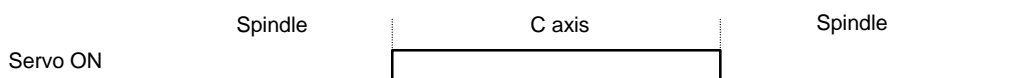
Since orient completion can be predicted using this function, it is possible to eliminate the sequence delay time, etc. for tool changes and other such operations, thereby achieving a faster tact time.

8.1.3.2 Spindle Position Control (Spindle / C Axis Control)

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	–	Δ	Δ	Δ	–	Δ	Δ	Δ
L system	–	Δ	Δ	Δ	–	Δ	Δ	Δ

This function enables one digital spindle (SP) to be used also as the C axis (rotary axis) using an external signal.

The C axis servo ON signal is used to switch between the spindle and C axis.



At servo OFF: ----- Spindle (C axis cannot be controlled).
C axis is in the reference position return incomplete status.

At servo ON: ----- C axis (spindle cannot be controlled).
C axis is in the reference position return incomplete status.
A parameter is used to initiate reference position return (orient) when servo ON is started.

8 Spindle, Tool and Miscellaneous Functions

8.1 Spindle Functions (S)

8.1.3.3 Spindle Synchronization

8.1.3.3.1 Spindle Synchronization I

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	-	-	-
L system	-	-	-	-	-	Δ	Δ	Δ

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).

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.

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 Spindle, Tool and Miscellaneous Functions

8.1 Spindle Functions (S)

8.1.3.3.2 Spindle Synchronization II

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	–	–	–	–	–	○	○	○
L system	–	–	–	–	–	Δ	Δ	Δ

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).

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.

The selection of the spindles to be synchronized, the start of the synchronization and other settings are all designated from the PLC.

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

8.1.3.4 Tool Spindle Synchronization I (Polygon)

8.1.3.4.1 Tool Spindle Synchronization IA (Spindle-Spindle, Polygon)

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	–	–	–	–	–	–	–	–
L system	–	Δ	–	–	–	Δ	Δ	Δ

With a machine equipped with two or more spindles under serial connection control, this function enables spindle-spindle polygonal machining (IA) by controlling the workpiece spindle rotation in synchronization with the rotary tool spindle rotation. The rotary tool spindle and workpiece spindle are designated among the spindles subject to serial connection control.

Serial connection control for the workpiece spindle and rotary tool spindle can be performed by MDS-*-SP or MDS-*-SPJ2.

Command format

Spindle-spindle polygonal machining mode command (G114.2)

This command establishes the spindle-spindle polygonal machining mode in which two spindles are rotated in synchronization at two different speeds by designating the rotary tool spindle and workpiece spindle and by designating the rotational ratios (number of rotary tool teeth and number of work angles) of the two designated spindles.

G114.2 H__ D__ E__ L__ R__ ;

H__ : Selects the rotary tool spindle.
D__ : Selects the workpiece spindle.
E__ : Designates the rotary tool spindle rotational ratio.
L__ : Designates the workpiece spindle rotational ratio.
R__ : Synchronized spindle phase shift amount

8 Spindle, Tool and Miscellaneous Functions

8.1 Spindle Functions (S)

Spindle-spindle polygonal machining mode cancel command (G113)

This command releases the state of synchronization between two spindles whose rotation has been synchronized by the spindle synchronization command.

G113;

8.1.3.4.2 Tool Spindle Synchronization IB (Spindle-Spindle, Polygon)

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	-	-	-
L system	-	-	Δ	Δ	-	Δ	Δ	Δ

With a machine equipped with two or more spindles under serial connection control, this function enables spindle-spindle polygonal machining (IB) by controlling the rotary tool spindle rotation in synchronization with the workpiece spindle rotation. The rotary tool spindle and workpiece spindle are designated among the spindles subject to serial connection control.

A parameter is used to switch from spindle-spindle polygonal machining (IB) to spindle-NC axis polygonal machining (IC) or vice versa.

Serial connection control for the workpiece spindle and rotary tool spindle can be performed by MDS-*-SP or MDS-*-SPJ2.

Command format

Spindle-spindle polygonal machining mode ON (G51.2 or G251)

This command establishes the spindle-spindle the polygonal machining mode in which two spindles are rotated in synchronization at two different speeds by designating the rotary tool spindle and workpiece spindle and by designating the rotational ratios (number of work angles and number of rotary tool teeth) of the two designated spindles.

G51.2 H__ D__ P__ Q__ R__ ;

H__ : Selects the workpiece spindle (spindle).
 D__ : Selects the rotary tool spindle (spindle).
 P__ : Designates the workpiece spindle rotational ratio.
 Q__ : Designates the rotary tool spindle rotational ratio.
 R__ : Synchronized spindle phase shift amount

Spindle-spindle polygonal machining cancel command (G50.2)

G50.2:

8 Spindle, Tool and Miscellaneous Functions

8.1 Spindle Functions (S)

8.1.3.4.3 Tool Spindle Synchronization IC (Spindle-NC axis, Polygon)

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	-	-	-
L system	-	-	Δ	Δ	-	Δ	Δ	Δ

With a machine equipped with a workpiece spindle subject to serial connection control and also an NC axis serving as the rotary tool axis, this function enables spindle-NC axis polygonal machining (IC) by controlling the rotary tool axis rotation in synchronization with the workpiece spindle rotation. A parameter is used to switch from spindle-spindle polygonal machining (IB) to spindle-NC axis polygonal machining (IC) or vice versa.

Serial connection control for the workpiece spindle can be performed by MDS*-SP or MDS*-SPJ2. The rotary tool axis can be performed by MDS*-V1/V2.

Command format

G51.2 P__ Q__ ;

P__ : Selects the workpiece spindle (spindle).

Q__ : Selects the rotary tool axis (NC axis).

Spindle-spindle polygonal machining cancel command (G50.2)

G50.2:

8 Spindle, Tool and Miscellaneous Functions

8.2 Tool Functions (T)

8.2 Tool Functions (T)

8.2.1 Tool Functions

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
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.

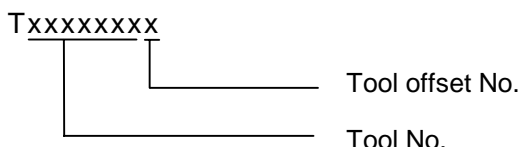
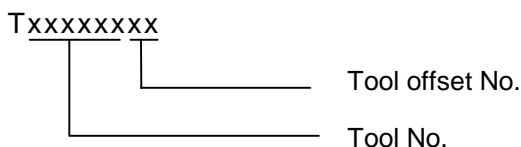
Processing and complete sequences must be incorporated on the PLC side for all T commands.

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

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

(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) This function requires a built-in PLC.

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

8 Spindle, Tool and Miscellaneous Functions

8.3 Miscellaneous Functions (M)

8.3 Miscellaneous Functions (M)

8.3.1 Miscellaneous Functions

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

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

M2-digit BCD output is used for the standard PLC specifications.

When a 2-digit number following address M (M00 – M97) is assigned, the code data and start signal will be 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 M00 to M97.

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/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

Four sets of M commands can be issued simultaneously in a block.

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

Respective processing and completion sequences are required for all M commands included in a block (except M98 and M99).

(Note 1) This function requires a built-in PLC. In this case, the code data and start signal are transferred simultaneously as the M command in the same block from the controller to the PLC, and so high-speed machine control can be done by the PLC processing sequence.

8 Spindle, Tool and Miscellaneous Functions

8.3 Miscellaneous Functions (M)

8.3.3 M Code Independent Output

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

When the M00, M01, M02 or M30 command is assigned during an automatic operation (tape, memory, MDI) or by a manual numerical command, the signal of this function is output. It is turned OFF after the miscellaneous function finishes or by the reset & rewind signal.

Machining program	M code output Independent	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

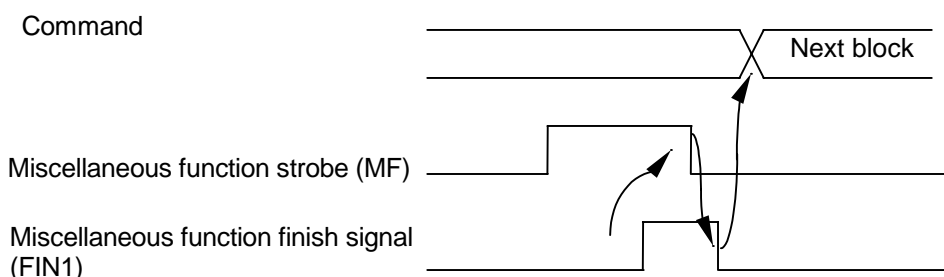
M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

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

Miscellaneous function finish signal 1 (FIN1)

When the controller checks that FIN1 is ON, it sets the function strobes OFF. Furthermore, when the PLC checks that the function strobes are OFF, it sets FIN1 OFF. The controller checks that FIN1 is OFF and advances to the next block.

Below is an example of a time chart applying when a miscellaneous function has been assigned.

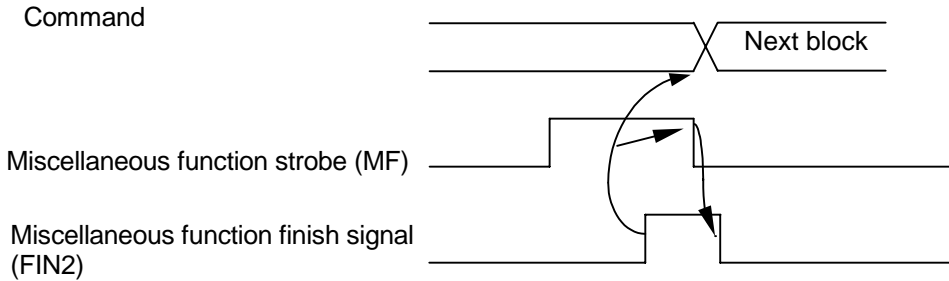


8 Spindle, Tool and Miscellaneous Functions

8.3 Miscellaneous Functions (M)

Miscellaneous function finish signal 2 (FIN2)

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



8.3.5 M Code Output during Axis Positioning

M/L system	Model	M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system		-	-	-	-	-	-	-	-
L system		-	-	-	-	-	Δ	Δ	Δ

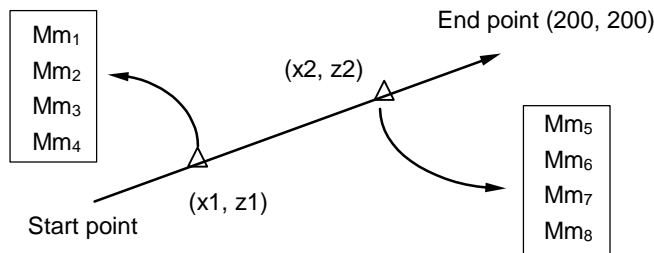
This function controls the timing at which miscellaneous functions are output, and it outputs a miscellaneous function when axis reaches at the designated position movement.

The command format is as follows.

G117 Xx1 Zz1 Cc1 □□□□ ;

G117 : In-axis-movement miscellaneous function output command
 Xx1, Zz1, Cc1 : Movement start points
 □□□□ : Miscellaneous function

(Example) G117 Xx₁ Zz₁ Mm₁ Mm₂ Mm₃ Mm₄ ;
 G117 Xx₂ Zz₂ Mm₅ Mm₆ Mm₇ Mm₈ ;
 G01 X200 Z200 ;



8 Spindle, Tool and Miscellaneous Functions

8.4 2nd Miscellaneous Functions (B)

8.4 2nd Miscellaneous Function (B)

8.4.1 2nd Miscellaneous Function

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
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) This function requires a built-in PLC.

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

9 Tool Compensation

9.1 Tool Length / Position Offset

9 Tool Compensation

9.1 Tool Length / Position Offset

9.1.1 Tool Length Offset

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

These commands make it possible to control the axis movement by offsetting the position of the end point of the movement command by an offset 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) When the tool length offset axis is returned to the reference position, 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
 $H01 = -450.000$.

→

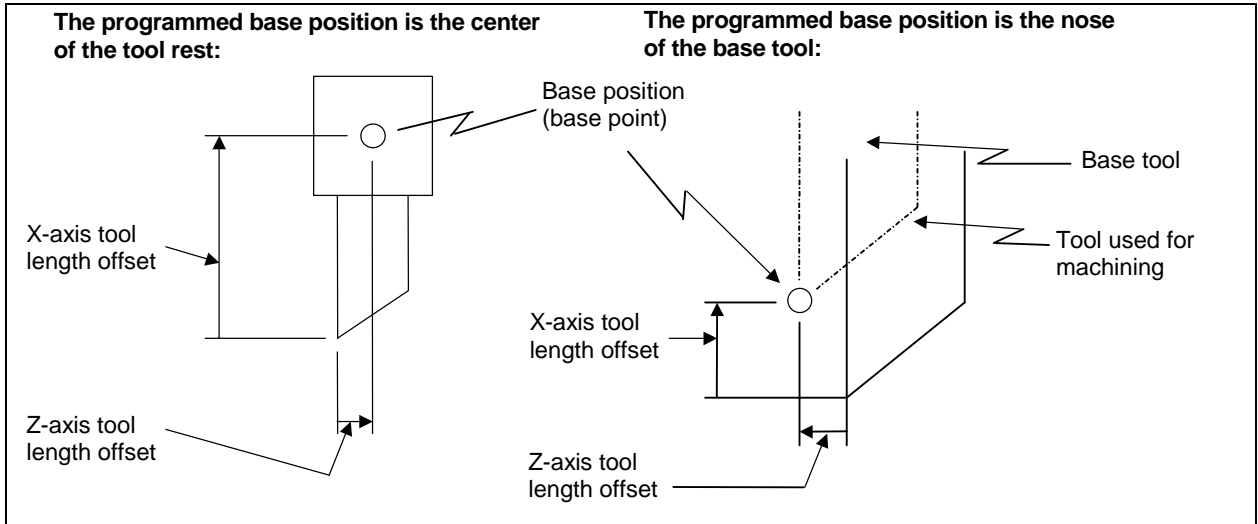
9 Tool Compensation

9.1 Tool Length / Position Offset

(2) L system

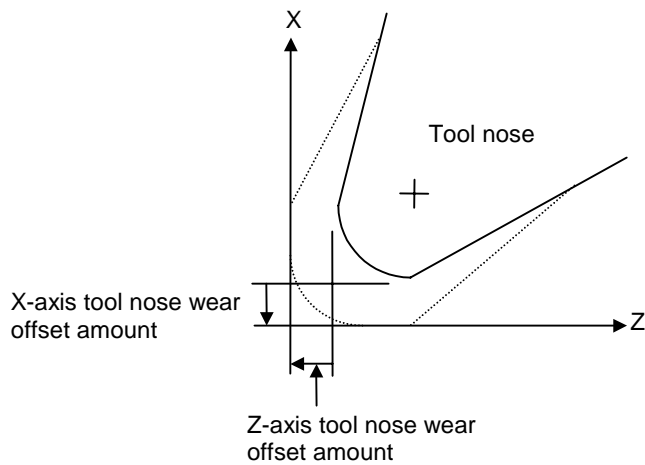
(a) Shape offset

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.



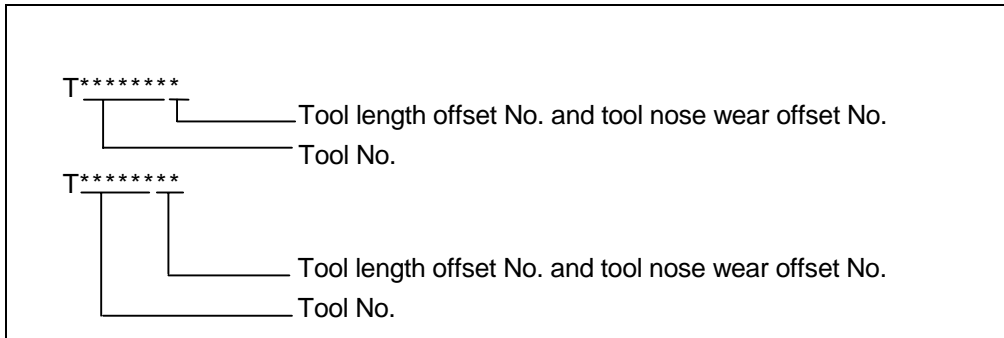
9 Tool Compensation

9.1 Tool Length / Position Offset

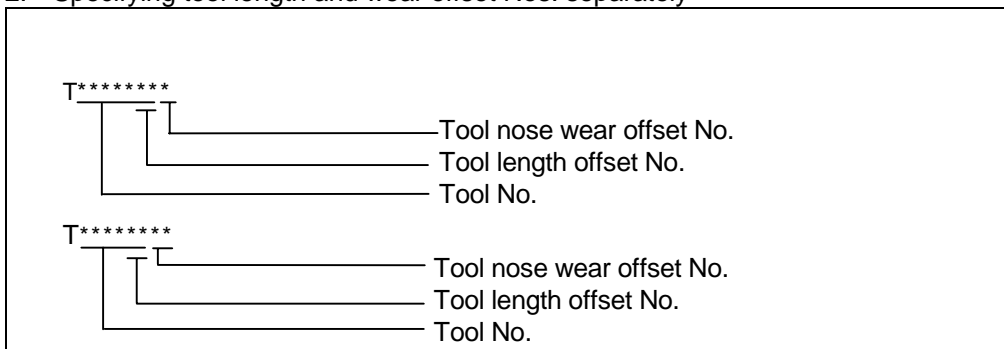
(c) Command format

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

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

9 Tool Compensation

9.1 Tool Length / Position Offset

9.1.2 Tool Position Offset

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	○	○	○	○
L system	-	-	-	-	-	-	-	-

This function uses commands to control the movement by changing the positions of the end points of the movement commands to positions which have been extended or reduced by an amount equivalent to the tool radius compensation amount.

This function can be used to compensate for the difference in distance between the position where the tool on the machine is actually mounted and the programmed coordinate position based on the tool position and thereby improve the efficiency of both machining and operation.

G45 G00 Xx1 Yy1 Dd1 ;	
G45	: Offset
Xx1, Yy1	: Movement axes
Dd1	: Offset No.

With tool position offset, the compensation operation is performed only for blocks containing a G45 to G48 command

G45 command	G46 command	G47 command	G48 command
Extension by amount equivalent to offset amount	Reduction by amount equivalent to offset amount	Extension by twice the offset amount	Reduction by twice the offset amount

- (1) If the start and end points are on an axis, the radius can be extended or reduced only for one-quarter, one-half and three-quarter arcs.
- (2) In the case of absolute value commands, the position is extended or reduced in each axial direction from the end point of the previous block along the line of the movement toward the position commanded in the block containing the G45 (or G46, G47 or G48) command.
- (3) In the case of simultaneous n axes command, the same amount of offset is applied to all the axes that have the command within the range of the number of the axes which can be simultaneously controlled. Tool position offset is also valid for additional axes.

9.1.3 Tool Offset for Additional Axes

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	-	-	-
L system	○	○	○	○	○	○	○	○

Tool compensation for the L series is valid for the X and Z axes. If an additional axis (such as the Y axis) has been added to these axes, tool compensation is valid for the additional axis.

9 Tool Compensation

9.2 Tool Radius

9.2 Tool Radius

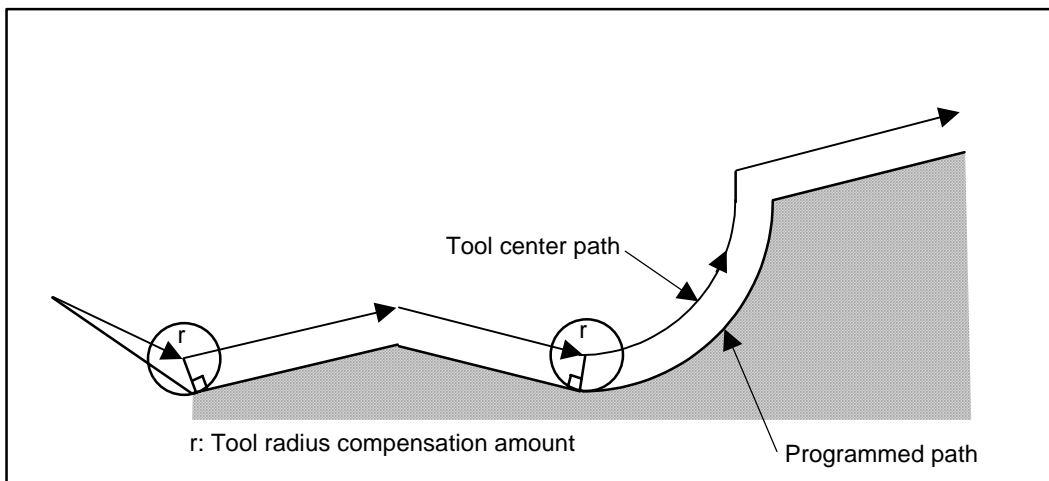
9.2.1 Tool Radius Compensation

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
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.

9 Tool Compensation

9.2 Tool Radius

G17 G01 G41 Xx1 Yy1 Dd1 ;

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

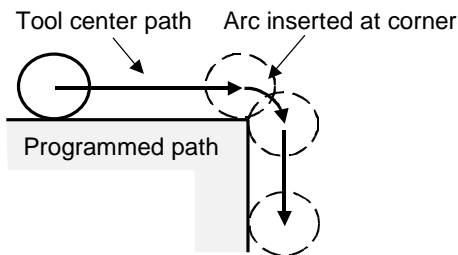
The compensation plane, movement axes and next advance direction vector are based on the plane selection command designated by G17 to G19.

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 ;

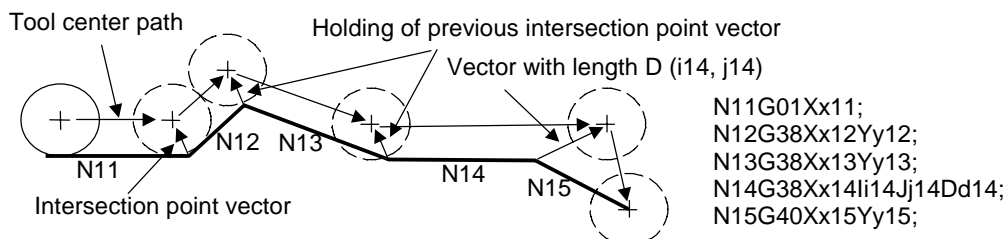
Xx1, Yy1 : Movement amount

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.



9 Tool Compensation

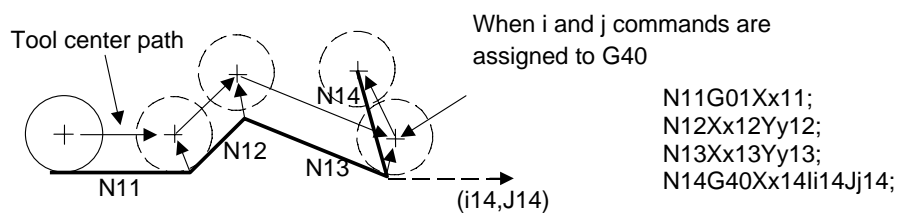
9.2 Tool Radius

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 Tool Compensation

9.2 Tool Radius

9.2.2 3-dimensional Tool Radius Compensation

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	Δ	Δ	-	-	Δ	Δ
L system	-	-	-	-	-	-	-	-

This command serves the function of compensating for the spherical radius of ball end mills. It corrects the actual tool center path to be either more outside or inside the programmed path by an amount equivalent to the tool radius amount in accordance with the 3-dimensional vectors.

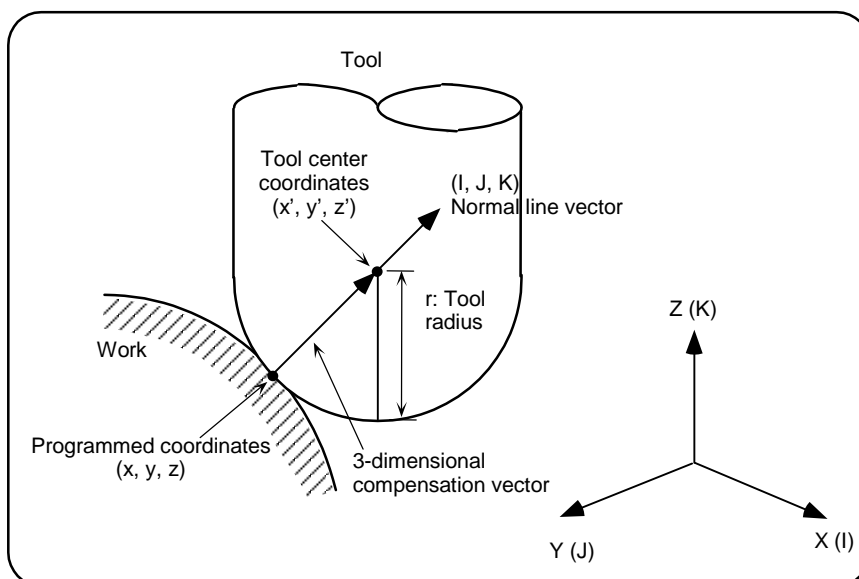
G code	Function
G40	Tool radius compensation cancel
G41	Tool radius compensation left command (compensation in the direction toward IJK)
G42	Tool radius compensation right command (compensation in the direction away from IJK)

With 3-dimensional tool radius compensation, the compensation is controlled from the block in which the block containing the G41 or G42 command has been designated.

G01 G41 Xx1 Yy1 Zz1 Ii1 Jj1 Kk1 Dd1 ;	
G01	: Cutting
G41	: Left compensation
Xx1, Yy1, Zz1	: Movement axis
Ii1, Jj1, Kk1	: Tool sphere center vectors
Dd1	: Offset No.

The radius from the tool spherical center to the end nose serves as the compensation amount that corresponds to the tool offset No. designated by d1. The offset Nos. that can be used are limited by the "number of tool compensation amount sets."

If the compensation vectors (tool spherical center vectors) are to be changed, the G41 and I, J and K commands must be assigned.



9 Tool Compensation

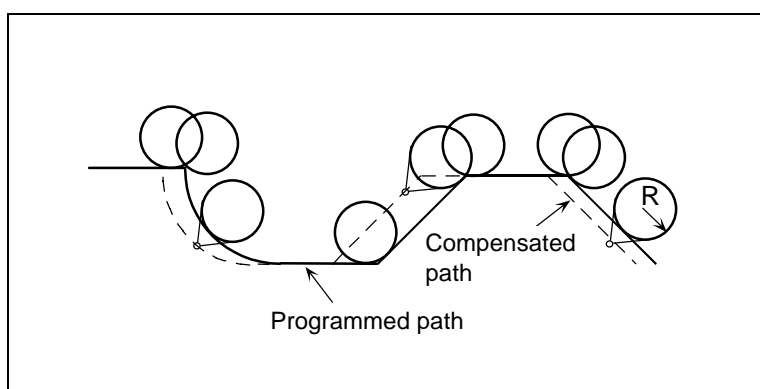
9.2 Tool Radius

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

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	-	-	-
L system	○	○	○	○	○	○	○	○

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

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



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 Tool Compensation

9.2 Tool Radius

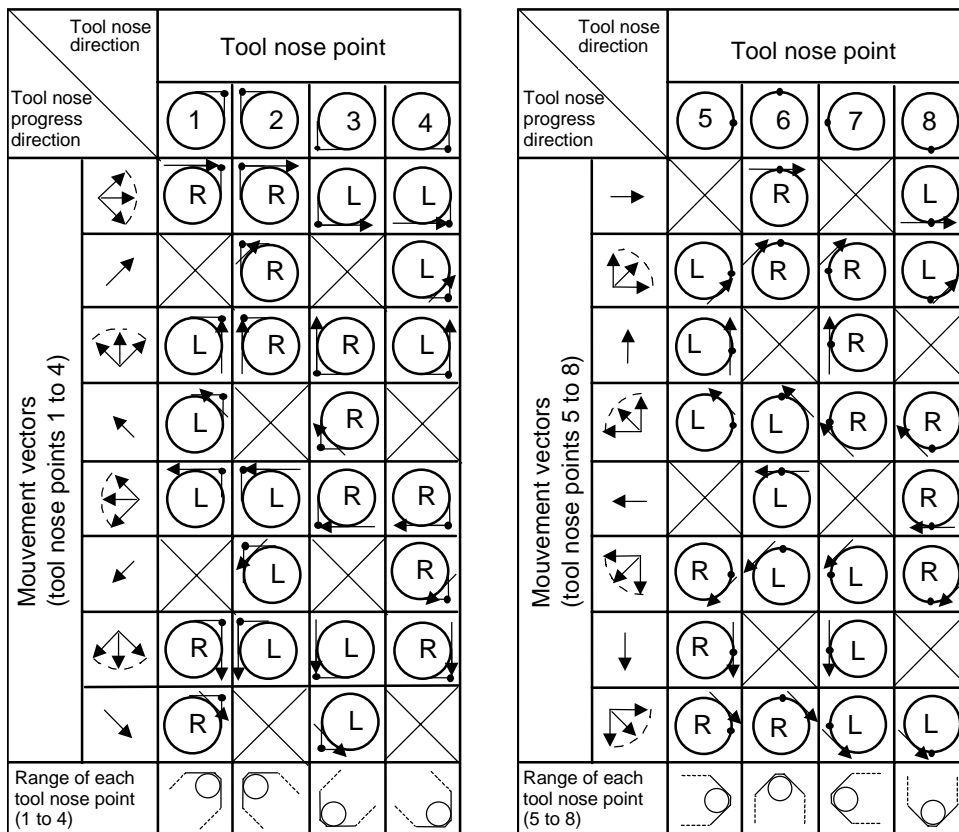
9.2.4 Automatic Decision of Nose Radius Compensation Direction (G46/40)

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	-	-	-
L system	○	○	○	○	○	○	○	○

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

G code	Function
G40	Nose radius compensation cancel
G46	Nose radius compensation ON (Automatic decision of compensation direction)

The compensation directions based on the movement vectors at the tool nose points are as follows:



9 Tool Compensation

9.3 Tool Offset Amount

9.3 Tool Offset Amount

9.3.1 Number of Tool Offset Sets

The number of tool offset sets is as follows.

9.3.1.1 20

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
	M system	-	-	-	-	-	-	-	-
	L system	○	○	○	○	○	○	○	○

9.3.1.2 40

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
	M system	○	○	○	○	○	○	○	○
	L system	-	-	-	-	-	-	-	-

9.3.1.3 80

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
	M system	-	-	-	-	-	-	-	-
	L system	△	△	△	△	△	△	△	△

9.3.1.5 200

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
	M system	△	△	△	△	△	△	△	△
	L system	-	-	-	-	-	-	-	-

9.3.1.6 400

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
	M system	-	△	-	-	-	△	△	△
	L system	-	-	-	-	-	-	-	-

9.3.1.7 999

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
	M system	-	-	-	-	-	-	△	△
	L system	-	-	-	-	-	-	-	-

9 Tool Compensation

9.3 Tool Offset Amount

9.3.2 Offset Memory

9.3.2.1 Tool Shape/Wear Offset Amount

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

This function registers the tool shape offset and wear offset amounts among the positions of the tools moving in the direction parallel to the control axis. 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.

9 Tool Compensation

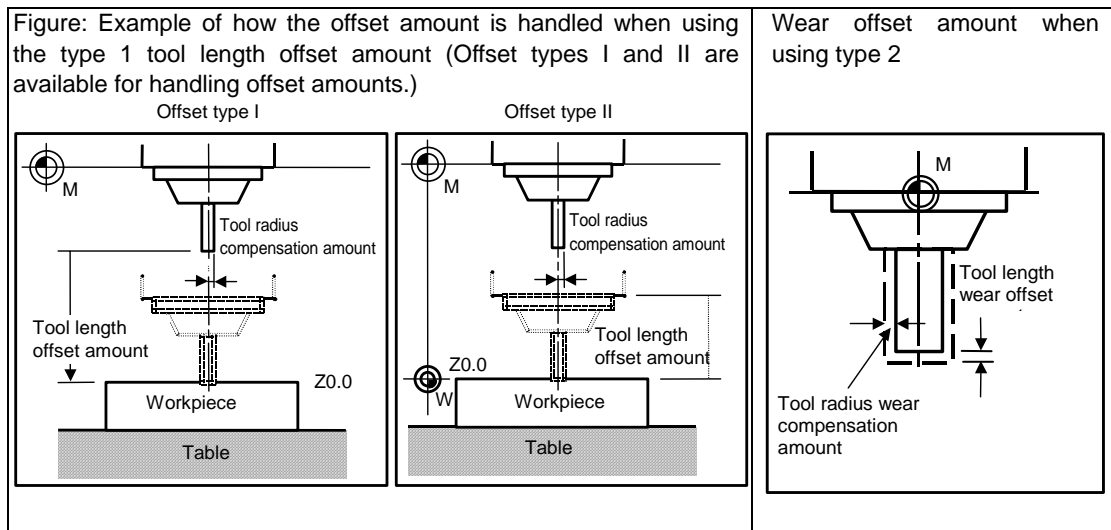
9.3 Tool Offset Amount

(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.



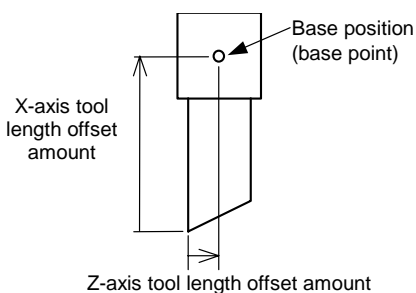
(c) Type 3: 2-axis offset amounts [L system]

Type 3 is for the offset amounts used by non-rotary tools.

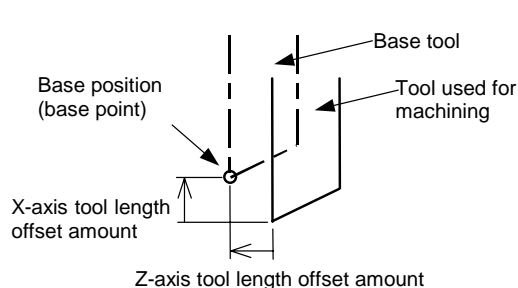
As the offset amounts, the tool length along the X, Y and Z axes and the wear amount along each of these axes, the nose radius and nose radius wear amount, tool tip point P and tool width can be registered.

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

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



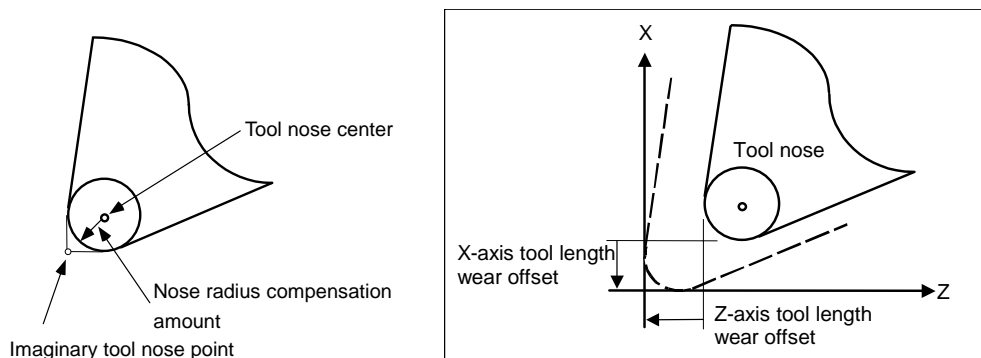
2. The programmed base position is the tip of the base tool:



9 Tool Compensation

9.3 Tool Offset Amount

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

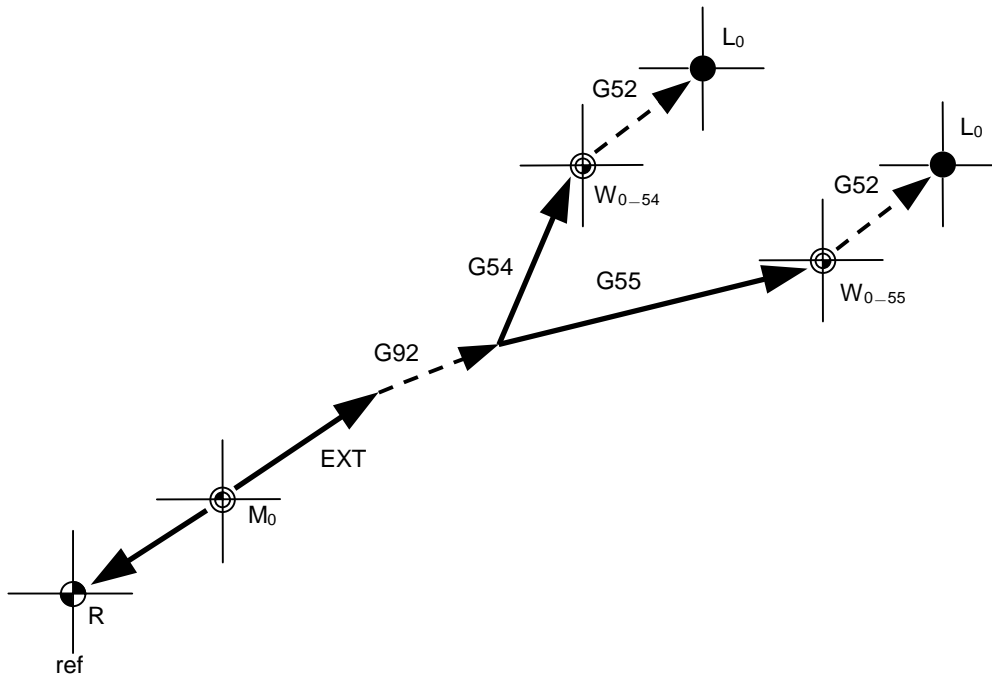
10.1 Coordinate System Type and Setting

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	\longrightarrow	Offset set with parameters
G52	Local coordinate system offset *1)	\dashrightarrow	Offset set with program
W_{0-54}	Workpiece coordinate system zero point (G54)		(0 when power is turned ON)
W_{0-55}	Workpiece coordinate system zero point (G55)		
G54	Workpiece coordinate system (G54) offset *1)		
G55	Workpiece coordinate system (G55) offset		
G92	G92 coordinate system shift		
EXT	External workpiece coordinate offset		
M_0	Machine coordinate system zero point		
ref	Reference position		

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

10 Coordinate System

10.1 Coordinate System Type and Setting

10.1.1 Machine Coordinate System

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

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

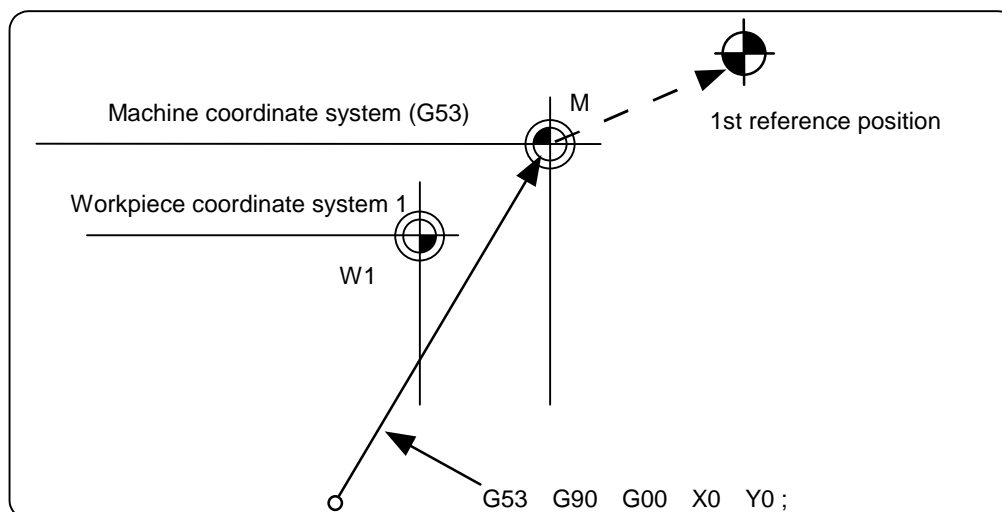
The programming format for the commands to move the tool to the machine coordinate system is given below.

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 Coordinate System

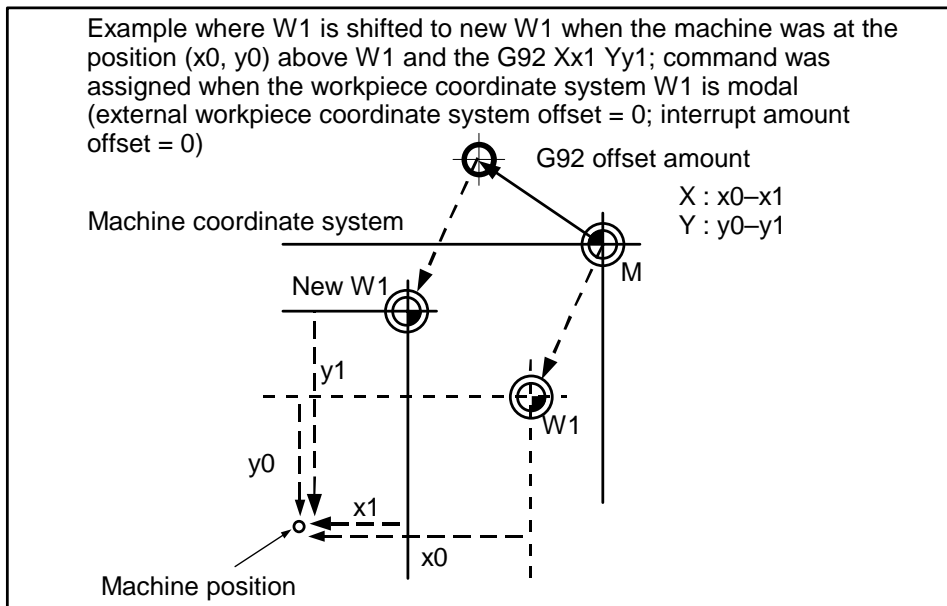
10.1 Coordinate System Type and Setting

10.1.2 Coordinate System Setting

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L 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 move, 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 system offset has been set) are shifted.

Offset of coordinate system by G92 coordinate system setting

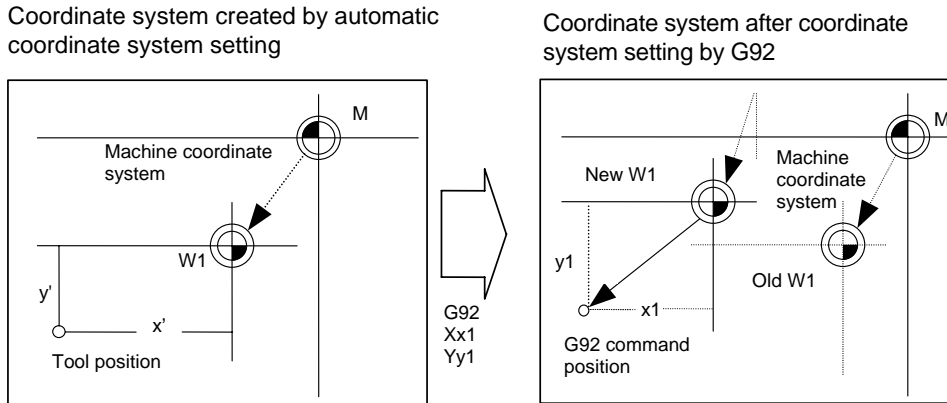


The shifted coordinate system is returned to its original position by dog-type reference position return or the program.

10 Coordinate System

10.1 Coordinate System Type and Setting

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.



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) Dog-type reference position return
 - b) Move to machine coordinate system zero point and assign G92 and G53 commands in same block to set the machine coordinate system.

<code>G90 G53 G00 X0 Y0 ;</code>	_____	Positioning at machine coordinate system zero point
<code>G92 G53 X0 Y0 ;</code>	_____	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/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

When the tool has arrived at the reference position by means of the first manual or automatic dog type reference position return after the controller power is turned ON, or immediately after the power is turned ON for the absolute position specifications, this function creates the coordinate systems in accordance with the parameters settings.

The coordinate systems created are given below.

1. Machine coordinate system corresponding to G53
2. G54 to G59 workpiece coordinate system
3. Local coordinate systems created under G54 to G59 workpiece coordinate systems

The distances from the zero point of G53 machine coordinate system are set to the controller coordinate related parameters.

10 Coordinate System

10.1 Coordinate System Type and Setting

10.1.4 Workpiece Coordinate System Selection (6 sets)

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

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

Up to 6 workpiece coordinate systems can be selected.

The G54 workpiece coordinate systems are 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.

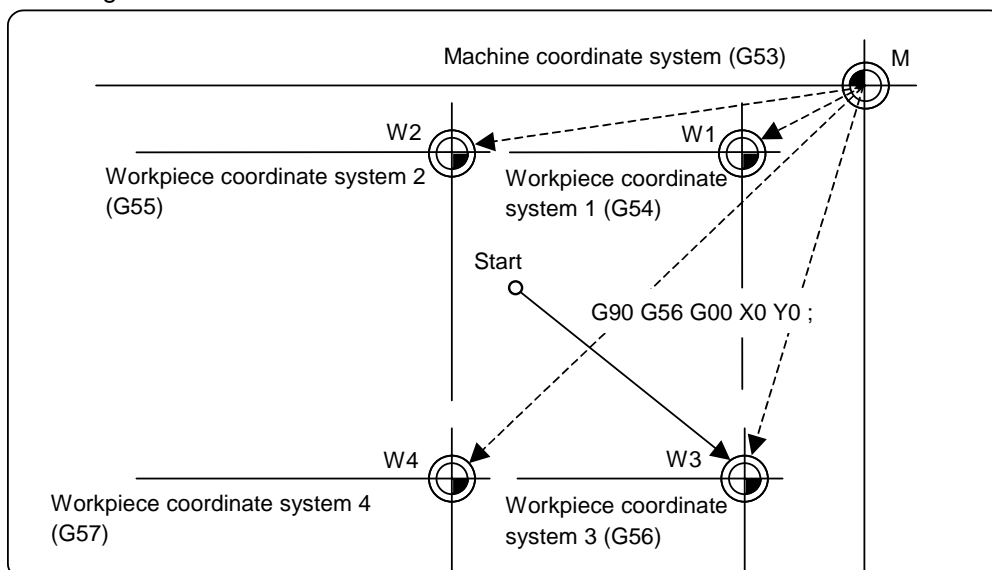
(G90) G54 G00 Xx1 Yy1 Zz1 ;

(G90) : (Absolute value command)
 G54 : Coordinate system selection
 G00 : Movement mode
 Xx1, Yy1, Zz1 : Coordinate positions of movement destination

The workpiece coordinate zero points are provided as distances from the zero point of the machine coordinate system.

Settings can be performed in one of the following three ways:

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



10 Coordinate System

10.1 Coordinate System Type and Setting

10.1.5 Extended Workpiece Coordinates System Selection (48 Sets) G54 1P1 to P48

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

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ

In addition to the six workpiece coordinate systems G54 to G59, 48 workpiece coordinate systems can be used by assigning G54.1Pn command.

The command format to select the workpiece coordinate system using the G54.1Pn command and to move on the workpiece coordinate system are given below.

<p>(G90) G54.1Pn G00 Xx1 Yy1 Zz1 ;</p> <p>(G90) : (Absolute value command) G54.1Pn : Coordinate system selection G00 : Movement mode Xx1, Yy1, Zz1 : Coordinate positions of movement destination</p>

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:

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

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

10 Coordinate System

10.1 Coordinate System Type and Setting

10.1.6 Workpiece Coordinate System Preset (G92.1)

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	-	-	-
L system	-	-	-	-	Δ	Δ	Δ	Δ

This function presets the workpiece coordinate system, which has been shifted by the programmed command or the manual operation, as the workpiece coordinate system which has been offset by the programmed command (G92.1) from the machine zero point by an amount equivalent to the workpiece coordinate offset amount.

The workpiece coordinate system is shifted from the machine coordinate system when the such operations or the programmed commands as below have been performed.

- When manual intervention has occurred in the manual absolute OFF status
- When a movement command was performed in the machine lock status
- When movement was initiated by handle interrupt
- When a movement command was performed in the mirror image mode
- When a local coordinate system was set using the G52 command
- When a workpiece coordinate system was shifted using the G92 command

Just as when manual reference position return has been performed, this function presets the workpiece coordinate system which has been shifted once to the workpiece coordinate system which has been offset from the machine zero point by an amount equivalent to the workpiece coordinate offset amount.

Furthermore, whether to preset relative coordinates as well is selected with a parameter.

Command format

G92.1 (G50.3) X0 Y0 Z0 α0 ; (where α is an additional axis)
--

Designate the addresses of the axes to be preset.

Axes whose addresses have not designated will not be preset.

Depending on the command type, G50.3 command is used in stead.

A program error results when a value other than 0 is commanded.

10 Coordinate System

10.1 Coordinate System Type and Setting

10.1.7 Local Coordinate System

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

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

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

G code	Function
G54 G52	Local coordinate system on the workpiece coordinate system 1
G55 G52	Local coordinate system on the workpiece coordinate system 2
G56 G52	Local coordinate system on the workpiece coordinate system 3
G57 G52	Local coordinate system on the workpiece coordinate system 4
G58 G52	Local coordinate system on the workpiece coordinate system 5
G59 G52	Local coordinate system on the workpiece coordinate system 6

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

(G54) G52 Xx1 Yy1 Zz1 ;

(G54) : Workpiece coordinate system selection

G52 : Local coordinate system setting

Xx1, Yy1, Zz1 : Local coordinate offset amount

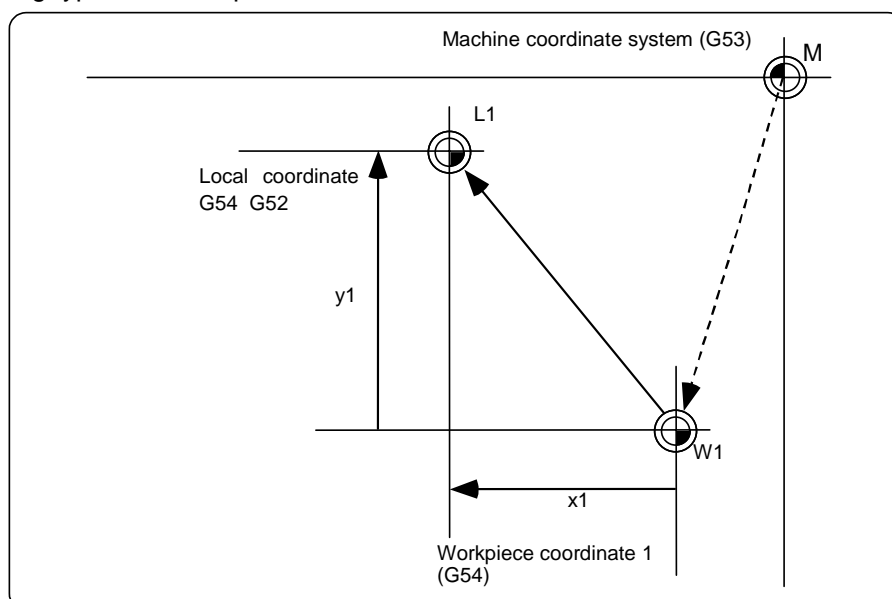
The local coordinate zero points are provided as distances from the zero point of the designated workpiece coordinate system (local coordinate offset).

In the incremental 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 position return.



10 Coordinate System

10.1 Coordinate System Type and Setting

10.1.8 Coordinate System for Rotary Axis

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

The coordinate system of rotary axis ranges from 0 to $\pm 360^\circ$. 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 $\pm 360^\circ$). 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/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

These G codes are for specifying the planes for the arc, tool radius compensation, coordinate rotation and other such commands.

G17 ; Xp-Yp plane designation
G18 ; Zp-Xp plane designation
G19 ; Yp-Zp plane designation

- (1) A parameter can be used to set either the X, Y or Z axis to which the additional axis is to be parallel.
- (2) A parameter can be used to set the initialization status (when the power has been turned ON or when the reset status has been entered) to G17, G18 or G19.
- (3) The movement commands have no connection with the plane selection.

Example

G19 X100. ; With these program commands, X100. is the axis which does not exist on the G19 (Yp, Zp) plane, Yp-Zp 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.

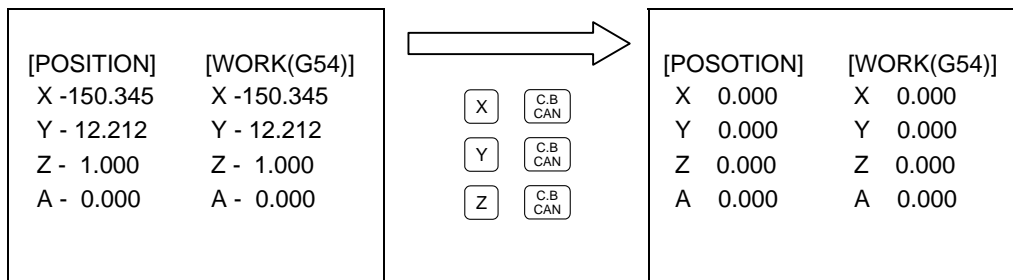
10 Coordinate System

10.1 Coordinate System Type and Setting

10.1.10 Origin Set

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

Using the setting and display unit, the coordinate system (current position and workpiece coordinate position) can be set to "0" by operating the screen. 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 key successively without pressing and keys.

10.1.11 Counter Set

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

Using the setting and display unit, the position counter display can be change to "0" by operating the screen.

- (1) This operation is the same as the operation of "Origin Set", but press key instead of key.
- (2) Only the [POSITION] counter display is changed to "0", and the other coordinate system counter displays are not changed.

10 Coordinate System

10.2 Return

10.2 Return

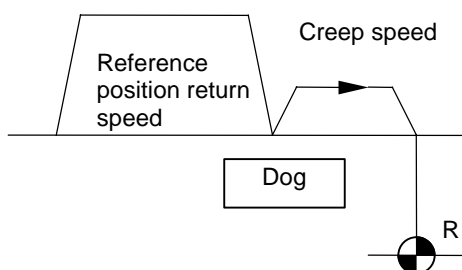
10.2.1 Manual Reference Position Return

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

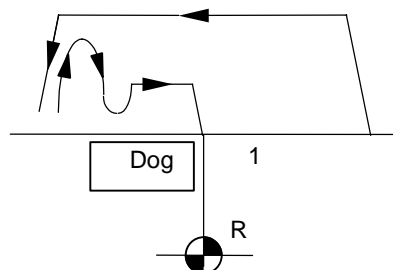
This function enables the tool to be returned manually to the position (reference position) which is characteristic to the machine.

(1) Return pattern to reference position

(a) Dog type

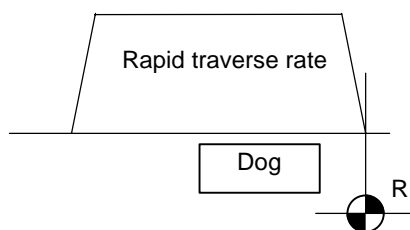


When starting in same direction as final advance direction



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
Absolute position detection method	High-speed	High-speed

10 Coordinate System

10.2 Return

10.2.2 Automatic 1st Reference Position Return

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

The machine can be returned to the first reference position by assigning the G28 command during automatic operation. If the interim point is commanded, the machine is moved up to that point by rapid traverse so that it is positioned and then returned separately for each axis to the first reference position.

Alternatively, by assigning the G29 command, the machine can be first positioned separately for each axis at the G28 or G30 interim point, and then positioned at the command position.

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

The G28 programming format is given below.

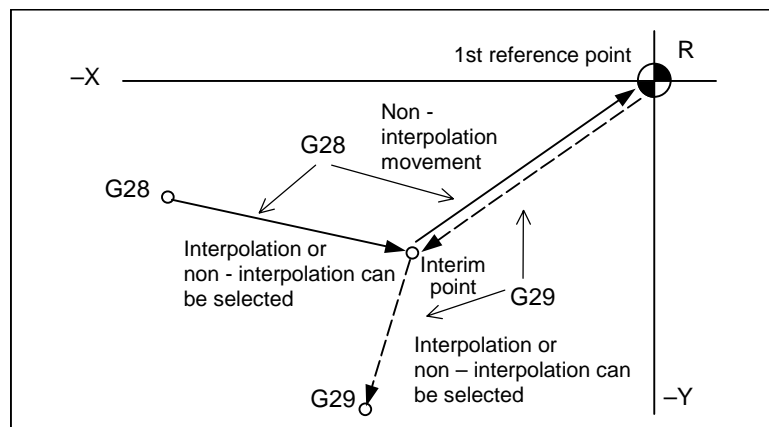
G28 Xx1 Yy1 Zz1 ;	
G28	: Return command
Xx1, Yy1, Zz1	: Return control axes (interim point)

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 position.

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.



10 Coordinate System

10.2 Return

If the position detector is for the incremental detection system, the first reference position return for the first time after the NC power has been turned ON will be the dog-type. However, whether the second and subsequent returns are to be the dog type or the high-speed type can be selected by designating a parameter.

The high-speed type is always used when the position detector is for the absolute position detection system.

- (Note 1)** The automatic 1st reference position return pattern is the same as for manual reference position return.
- (Note 2)** The number of axes for which reference position return can be performed simultaneously depends on the number of simultaneously controlled axes.
- (Note 3)** If, at the time of the first reference position return, the tool radius compensation or nose radius compensation has not been canceled, it will be temporarily canceled 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 position 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 position, 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 position 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 Coordinate System

10.2 Return

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

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

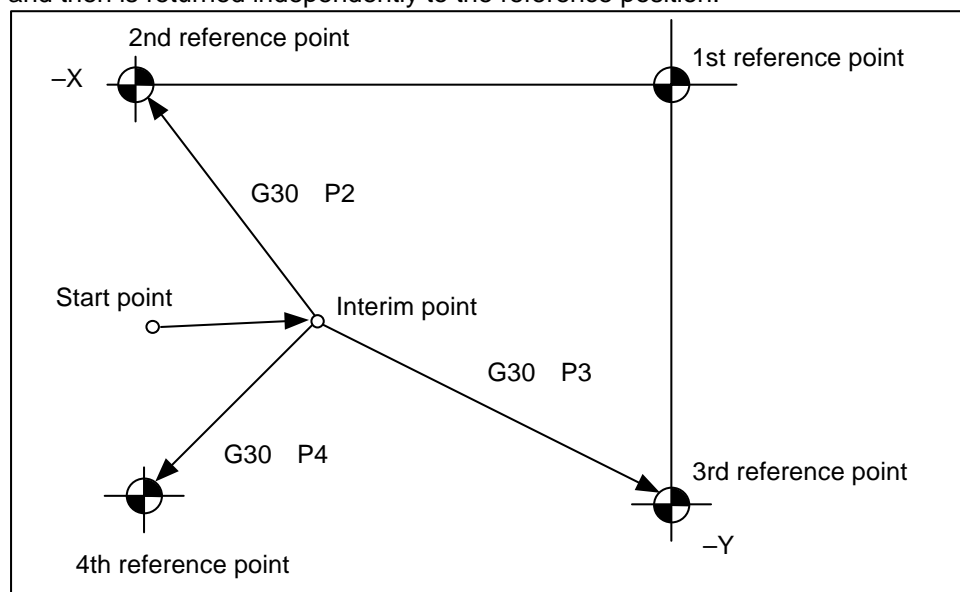
As with automatic 1st reference position return, commanding G30Pn during automatic operation enables the tool to be returned to the set points (2nd, 3rd or 4th reference positions) characteristic to the machine. The 2nd, 3rd and 4th reference positions can be set by parameters.

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

The G30 programming format is given below.

G30 Xx1 Yy1 Zz1 Pp1 ;	
G30	: Return command
Xx1, Yy1, Zz1	: Return control axes (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 position.



- (Note 1)** The second reference position return is performed if the P address is omitted.
- (Note 2)** The number of axes for which reference position return can be performed simultaneously depends on the number of simultaneously controlled axes.
- (Note 3)** If, at the time of the reference position return, the tool radius compensation 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.

10 Coordinate System

10.2 Return

(Note 4) If, at the time of the reference position return, the tool length offset has not been canceled, it will be canceled and the offset amount also cleared upon completion of reference position 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 positions.

(Note 6) The machine will not stop at the interim point even when a single block is selected.

10.2.4 Reference Position Verification

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

By commanding G27, a machining program, which has been prepared so that the tool starts off from the reference position and returns to the reference position, can be checked to see whether the tool will return properly to the reference position.

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 position verification

P2 : 2nd reference position verification

P3 : 3rd reference position verification

P4 : 4th reference position verification

The assigned axis is first positioned by rapid traverse to the commanded position and then, if this is the reference position, the reference position arrival signal is output.

When the address P is omitted, the first reference position verification will be applied.

(Note 1) The number of axes for which reference position 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 position upon completion of the command.

(Note 3) Whether interpolation or non-interpolation is to apply to the movement can be selected using a parameter.

10 Coordinate System

10.2 Return

10.2.5 Absolute Position Detection

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	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 position return. (High-speed return will always be used for the reference position 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-type		Same method as incremental detection dog-type	Zero point is established with dog-type reference position return completion.	The data is set in the parameter of zero point shift amount.
Dog-less type	Marked point method	The zero point position is set from the screen.	The zero point is established by input from the zero point initialization screen.	The value equivalent to the shift amount is set in the zero point initialization screen.
	Machine stopper method	The zero point is established by pressing the machine against a set point on the machine.	The zero point is established when a torque limit is applied on the servo and the torque limit is reached by pressing against the machine stopper.	The value equivalent to the shift amount is set in the zero point initialization screen.

Diagnosis during absolute position detection

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

10 Coordinate System

10.2 Return

10.2.6 Tool Exchange Position Return

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

By specifying the tool change position in a parameter and also assigning a tool exchange position return command in a machining program, the tool can be changed at the most appropriate position. The axes for which returning to the tool change position is performed and the order in which the axes begin to return can be changed by commands.

G30.n ;
 n = 1 to 6 : Specify the axes that return to the tool change position and the order in which they return. (For L system, n = 1 to 5)

Command and return order

[M system]

Command	Return order
G30.1	Z axis → X axis • Y axis (→ additional axis)
G30.2	Z axis → X axis → Y axis (→ additional axis)
G30.3	Z axis → Y axis → X axis (→ additional axis)
G30.4	X axis → Y axis • Z axis (→ additional axis)
G30.5	Y axis → X axis • Z axis (→ additional axis)
G30.6	X axis • Y axis • Z axis (→ additional axis)

[L system]

Command	Return order
G30.1	X axis only (→ additional axis)
G30.2	Z axis only (→ additional axis)
G30.3	X axis → Z axis (→ additional axis)
G30.4	Z axis → X axis (→ additional axis)
G30.5	X axis • Z axis (→ additional axis)

(Note1) An arrow (→) indicates the order of axes that begin to return. A period (•) indicates that the axes begin to return simultaneously.

Example : "Z axis → X axis" indicate that the Z axis returns to the tool change position, then the X axis does.

(Note2) G30.6 is only for the M system.

The tool exchange position return ON/OFF for the additional axis can be set with parameter for the additional axis. For the order to return to the tool change position, the axes return after the standard axis completes the return to the tool change position (refer to above table).

The additional axis cannot return to the tool change position alone.

10 Coordinate System

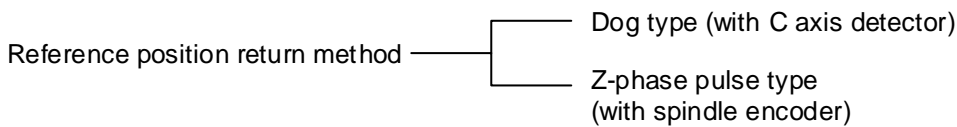
10.2 Return

10.2.7 C Axis Reference Position Return

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	-	-	-
L system	-	-	-	-	Δ	Δ	Δ	Δ

This function is used to carry out the position control for the spindle with the axis motor, and applied to the machine which can switch the spindle motor connected with the spindle to the axis motor. The C axis (rotation axis) is generally used for the axis motor, and the specification will be the same as the normal C axis control after switching.

As for the reference position return during the C axis connection, either the normal dog-type reference position return with the C axis or the Z-phase pulse type reference position return with the spindle encoder can be selected with parameter.



(Supplements)

(1) Reference position return method

The Z-phase pulse type is applied in the first reference position return after the servo OFF for the C axis (generally means changing to the spindle). The high-speed type is applied in the (second or later) reference position return after the Z-phase pulse type reference position return.

(2) Z-phase pulse type reference position return

When there is the reference position return command, the spindle is rotated until the Z-phase pulse of the spindle encoder is detected, and then stopped. (Figure 1 (1))

Encoder gear ratio 1:1 The Z-pulse is detected within one rotation.

Encoder gear ratio 1:2 The Z-pulse is detected within two rotations.

Next, in order to catch the change of the pulse position and improve the accuracy of the remaining distance, the spindle is rotated with G28crp (approach speed) and then stopped again. (Figure 1 (2))

Set as follows: Standard value G28crp = 8 (°/min) (Encoder gear ratio 1:1)

For the spindle, one rotation to the point where the Z-pulse is detected is made with G28rap (G28 rapid traverse rate). (Figure 1 (3)) This point is applied to the reference position. If the reference position shift amount is included, the spindle is rotated to the point of the figure 1 (4) and then stopped.

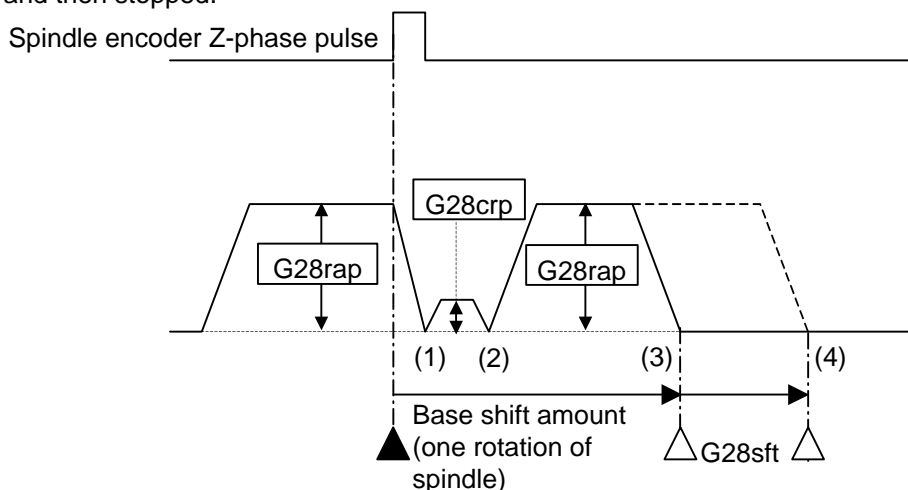


Figure 1 Z-phase pulse type reference position return

10 Coordinate System

10.2 Return

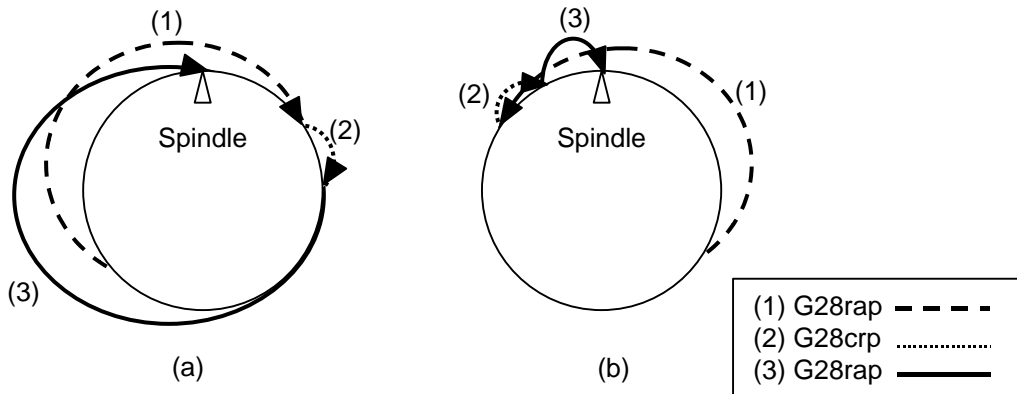


Figure 2 Operation of spindle

With the figure 2 (a), the rotation of the spindle that the G28 rapid traverse is performed in the forward direction (the same direction as the parameter #2030) is looked from the front. With the figure 2 (b), the rotation of the spindle that the G28 rapid traverse is performed in the opposite direction (the opposite direction to the parameter #2030) is looked from the front.

(3) High-speed type reference position return

After the Z-phase pulse type reference position return, the high-speed type reference position return is applied in the second or later reference position return.

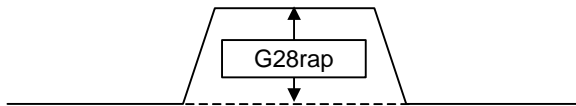


Figure 3 High-speed type reference position return

11 Operation Support Functions

11.1 Program Control



11 Operation Support Functions

11.1 Program Control

11.1.1 Optional Block Skip

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

When "/" (slant 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.

Programming example	Optional block skip	
	 Switch OFF	 Switch ON
N1	N1	N1
N2	N2	N2
N3	N3	N3
/N4	N4	<div style="border: 1px dashed black; height: 20px; width: 100%;"></div>
/N5	N5	<div style="border: 1px dashed black; height: 20px; width: 100%;"></div>
N6	N6	N6
N7	N7	N7
:	:	:

11 Operation Support Functions

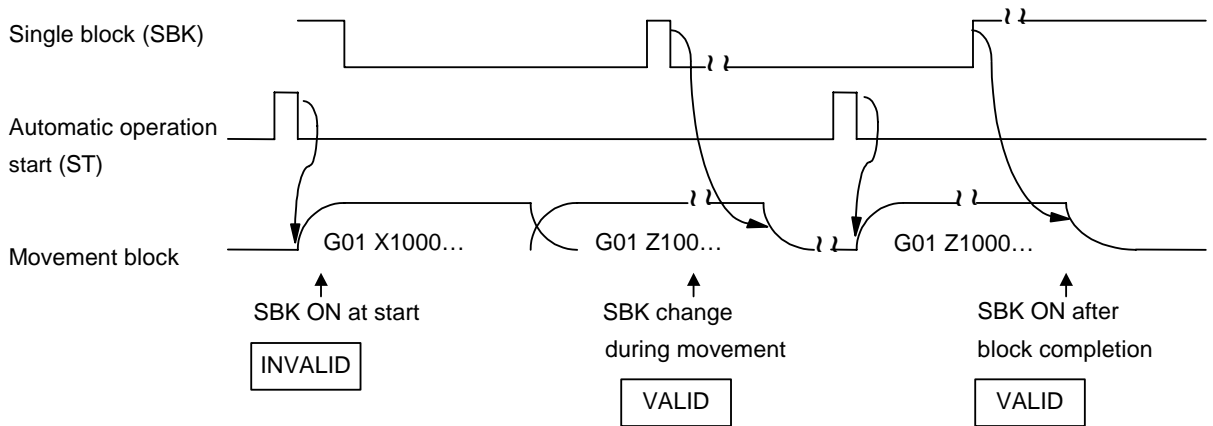
11.1 Program Control

11.1.3 Single Block

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

The commands for automatic operation can be executed one block at a time (block stop) by turning ON the single block input signal. When the single block input signal is turned ON temporarily during continuous operation, the machine will stop after that block has been executed.

When operation is switched to another automatic operation mode (for example, memory operation mode to MDI operation mode) during continuous operation, the machine will stop after that block has been executed.



As with the multi-part system specification, the following function can be selected instead.

Multi-part system single block (L system)

This function is for executing single block operation while maintaining the synchronization between the part systems when two or more part systems are operated. When one part system has been stopped by single block stop, the other part systems are stopped by feed hold.

11 Operation Support Functions

11.2 Program Test

11.2 Program Test

11.2.1 Dry Run

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
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	Manual rapid traverse selector switch OFF	Manual rapid traverse selector switch ON
G00, G27, G28, G29, G30, G60	Manual feed rate	Rapid traverse rate
G01, G02, G03	Manual feed rate	Cutting clamp speed

11.2.2 Machine Lock

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

When the machine lock input signal is set to ON, the NC operations can be executed without assigning commands to the NC axes.

The feed rate in the machine lock status is the command speed.

The M, S, T and B commands are executed as usual and operation is completed by returning the FIN signal.

- (1) Reference position return (manual, G28, G29, G30) is controlled as far as the interim point in the machine lock status but the block is completed when the interim point is reached.
- (2) Machine lock is effective in the signal status applying when the axis has stopped.
- (3) Block stop will be applied if the machine lock signal is turned ON and OFF or OFF and ON during automatic operation.

All axes will be simultaneous with the standard PLC.

11 Operation Support Functions

11.2 Program Test

11.2.3 Miscellaneous Function Lock

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		○	○	○	○	○	○	○	○
M system		○	○	○	○	○	○	○	○
L system		○	○	○	○	○	○	○	○

The M, S, T and B (2nd miscellaneous function) output signals are not output to the machine or PLC when the miscellaneous function lock signal of external input is turned ON. This function can be used when checking only the movement commands in a program check.

The start signals of the M command are output for the M00, M01, M02 and M30 commands, and so a completion signal must be returned.

- (1) Fixed cycle spindle functions containing an S code and any M, S, T or B function assigned by a manual numerical command or in automatic operation will not be executed. The code data and strobe (MF, SF, TF, BF) outputs are stopped.
- (2) If this signal is set ON after the code data has already been output, the output is executed as it would normally be executed until the end (until FIN1 or FIN2 is received and the strobe is turned OFF).
- (3) Even when this signal is ON, the M00, M01, M02 and M30 commands among the miscellaneous functions are executed, and the decode signal, code data and strobe signals are also output as they would be normally.
- (4) Any miscellaneous functions which are executed only inside the controller and not output (M96, M97, M98, M99) are executed as they would be normally even if this signal is ON.

11.2.4 Graphic Check

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		△	△	△	△	△	△	△	△
M system		△	△	△	△	△	△	△	△
L system		△	△	△	△	△	△	△	△

The movement path of the machine tool can be monitored and traced, and the path of machining programs can be traced and checked using the check and tracing functions based on processing inside the controller.

This function enables the tool path of machining programs to be traced without operating any functions.

For the display mode, 1-plane, 2-plane and 3-dimensional display are provided. In the 3-dimensional display mode, cubic shapes can be rotated and tracing of the figure seen from the desired direction can be assigned.

11.2.5 Graphic Trace

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		△	△	△	△	△	△	△	△
M system		△	△	△	△	△	△	△	△
L system		△	△	△	△	△	△	△	△

The machine position of the machine tool is traced. By this operation, the actual movement path in automatic operation or manual operation is traced.

For the display mode, 1-plane, 2-plane and 3-dimensional display are provided. In the 3-dimensional display mode, cubic shapes can be rotated and tracing of the figure seen from the desired direction can be assigned.

11 Operation Support Functions

11.3 Program Search / Start / Stop

11.3 Program Search / Start / Stop

11.3.1 Program Search

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

The program No. of the program to be operated automatically can be designated and called. Upon completion of search, the head of the program searched is displayed. Machining programs are stored in the memory inside the NC system.

11.3.2 Sequence Number Search

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

Blocks can be indexed by setting the program No., sequence No. and block No. of the program to be operated automatically.

The searched program is displayed upon completion of the search. Machining programs are stored in the memory inside the NC system.

11 Operation Support Functions

11.3 Program Search / Start / Stop

11.3.3 Collation Stop

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	Δ	Δ	Δ	Δ
L system	-	-	-	-	Δ	Δ	Δ	Δ

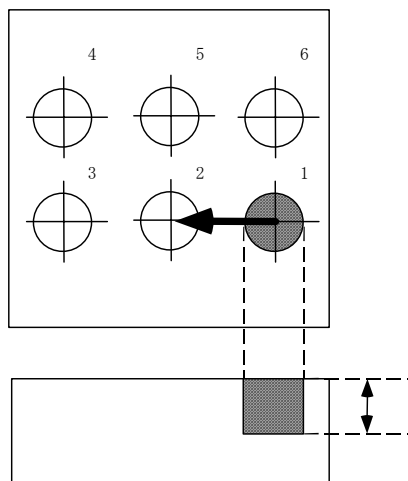
This function enables the single block stop status to be established at any block without having to set the SINGLE BLOCK switch to ON.

It can be used to readily check the machining shape up to the designated block and resume machining.

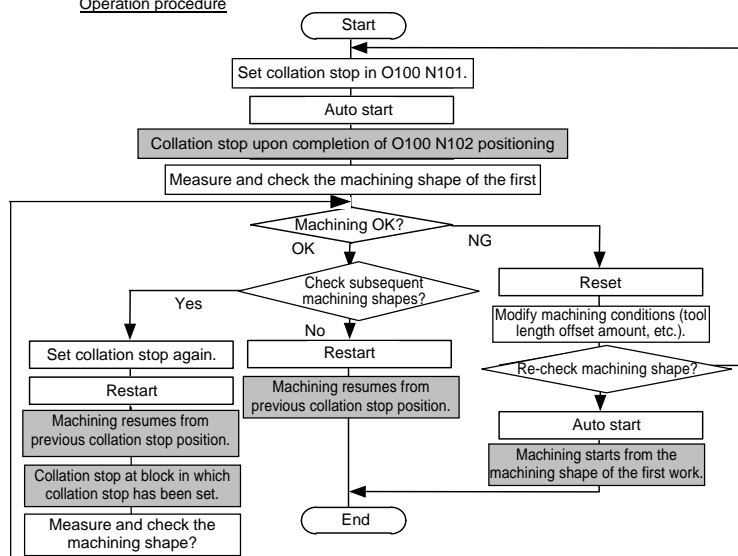
(Example)

```

O100:
    G91;
    .....
    G00 Z-150.;
▶ N100 G81 X-100. Z-100. R-50. F100;
    N101 X-100.
    N102 X-100.
    N103 Y100.
    N104 X100.
    N105 X100.
    .....
    .....
    
```



Operation procedure



11 Operation Support Functions

11.3 Program Search / Start / Stop

11.3.4 Program Restart

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	-	-	-	-	Δ	Δ	Δ	Δ

When a machining program is to be resumed after it has been suspended midway due to tool damage or for some other reason, this function searches the program and the block to be resumed and enables machining to be resumed from the block. When multiple part systems are used, only for 1-part system, the program can be resumed.

There are two resumption methods, type 1 and type 2.

Resumption type 1

Machining is resumed by type 1 if feed hold has been performed due to tool damage, etc. or if resetting has been performed.

(a) Type A (standard specification)

The designated sequence No. and block No. are searched only in the designated program No.. In the case of the standard specification, the program No. cannot be omitted.

(b) Type B

The program No. cannot be input since the main program which has been searched itself serves as the target. (A setting error results if the program No. is input.)

The designated sequence No. and block No. are searched in all the programs (including sub-programs) among the searched programs.

Resumption type 2

(a) Type A (standard specification)

If, before a resume search is initiated for the machining program to be resumed, a machining program differing from that program was run in the tape or memory mode, the machining program to be resumed will be resumed using type 2. It is also resumed using type 2 in cases where the coordinate system to be used when machining is resumed is to be changed from the coordinate system used during the previous automatic operation.

The operation sequence for type 2 is the same as for type 1. However, the coordinate system settings and other operations which must be performed before running the machining program must all be performed before initiating the resume search. The main program to be resumed should be searched at any time up to the moment immediately prior to starting the resumption of the machining.

The designated sequence No. and block No. are searched only in the designated program No.. In the case of the standard specification, the program No. cannot be omitted.

(b) Type B

If, before a resume search is initiated for the machining program to be resumed, a machining program differing from that program was run in the tape or memory mode, the machining program to be resumed will be resumed using type 2. It is also resumed using type 2 in cases where the coordinate system to be used when machining is resumed is to be changed from the coordinate system used during the previous automatic operation.

The operation sequence for type 2 is the same as for type 1. However, the coordinate system settings and other operations which must be performed before running the machining program must all be performed before initiating the resume search.

The program No. cannot be input since the main program which has been searched itself serves as the target. (A setting error results if the program No. is input.)

Therefore, the main program to be resumed must be searched before the resume search is initiated. The designated sequence No. and block No. are searched in all the programs (including sub-programs) among the searched programs.

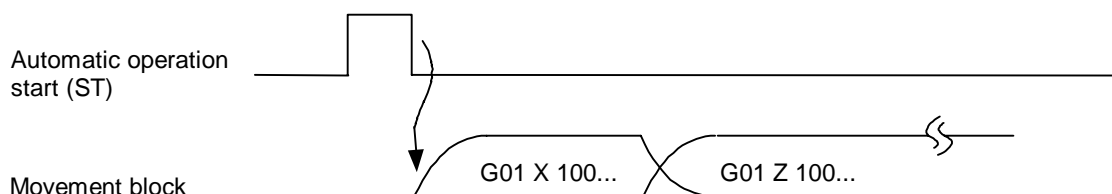
11 Operation Support Functions

11.3 Program Search / Start / Stop

11.3.5 Automatic Operation Start

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

With the input of the automatic operation start signal (change from ON to OFF), the automatic operation of the program that has been operation searched is started by the controller (or the halted program is restarted).



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

11.3.6 NC Reset

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

This function enables the controller to be reset.

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

11 Operation Support Functions

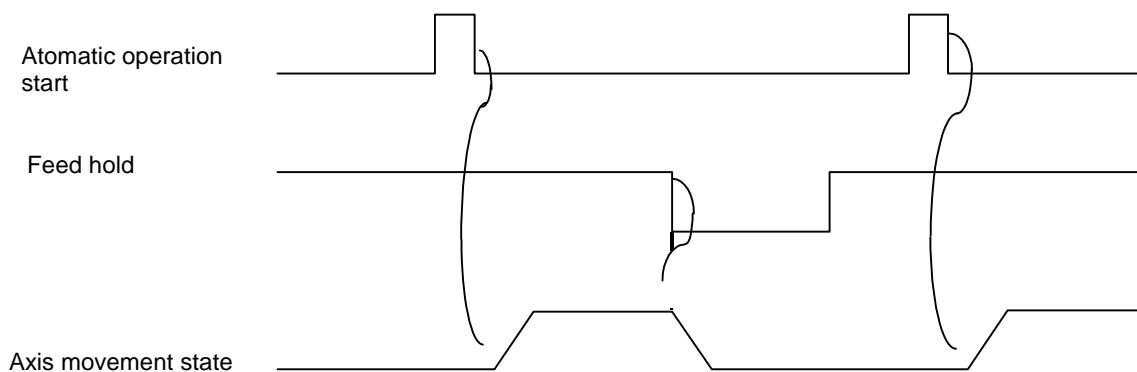
11.3 Program Search / Start / Stop

11.3.7 Feed Hold

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

When the feed hold signal is set ON during automatic operation, the machine feed is immediately decelerated and stopped. The machine is started again by the "Automatic operation start (cycle start)" signal.

- (1) When the feed hold mode is entered during automatic start, the machine feed is stopped immediately, but the M, S, T and B commands in the same block are still executed as programmed.
- (2) When the mode is switched during automatic operation to manual operation (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/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

If the search & start signal is input in a status where the memory mode is selected, the designated machining program is searched and executed from its head.
 If the search & start signal has been input during automatic operation in the memory mode, search & start is executed after resetting.

11 Operation Support Functions

11.4 Interrupt Operation

11.4 Interrupt Operation

11.4.1 Manual Interruption

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

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 Operation Support Functions

11.4 Interrupt Operation

11.4.2 Automatic Operation Handle Interruption

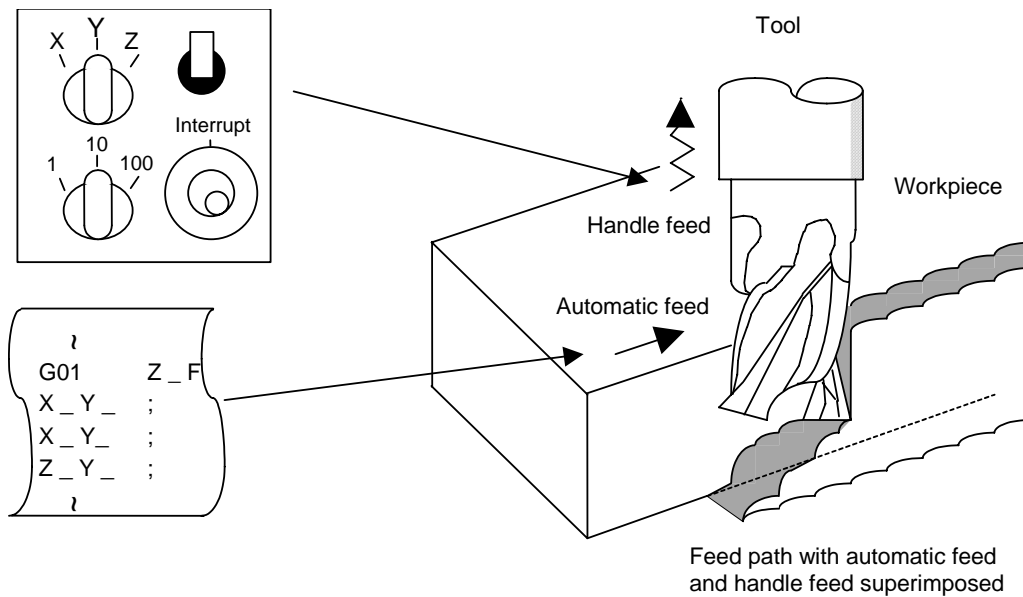
M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

The handle command can interrupt and be superimposed onto a command without suspending automatic operation and the machine can be moved by rotating the manual pulse generator during automatic operation.

If the spindle load is greatly exceeded when cutting a workpiece as per the machining program due to a high rough cutting amount in face machining, for instance, automatic handle interrupt makes it possible to raise the Z surface and reduce the load easily without suspending feed in the automatic operation mode.

Automatic handle interrupt is conducted by setting the "automatic handle interrupt" valid switch which is provided separately from the "manual operation mode". The axis selection 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.



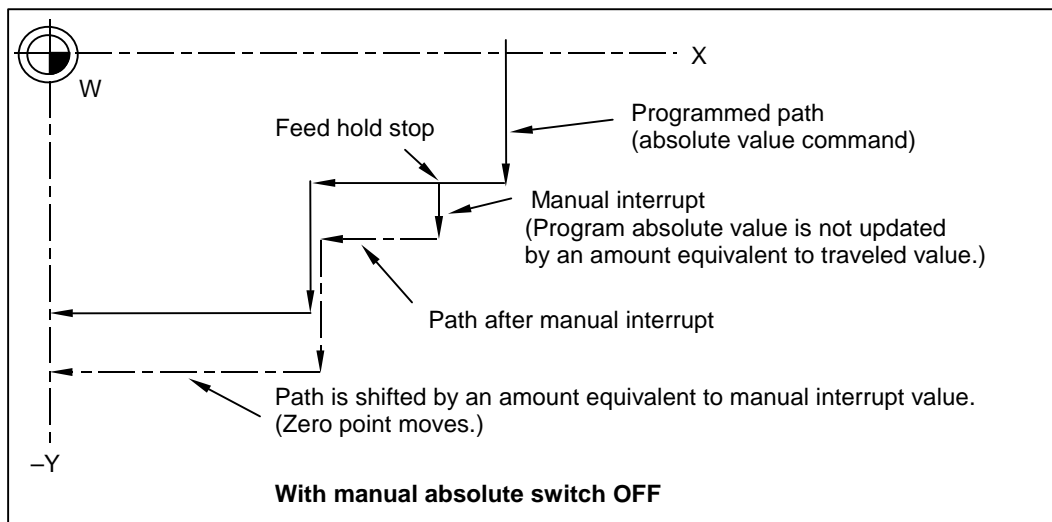
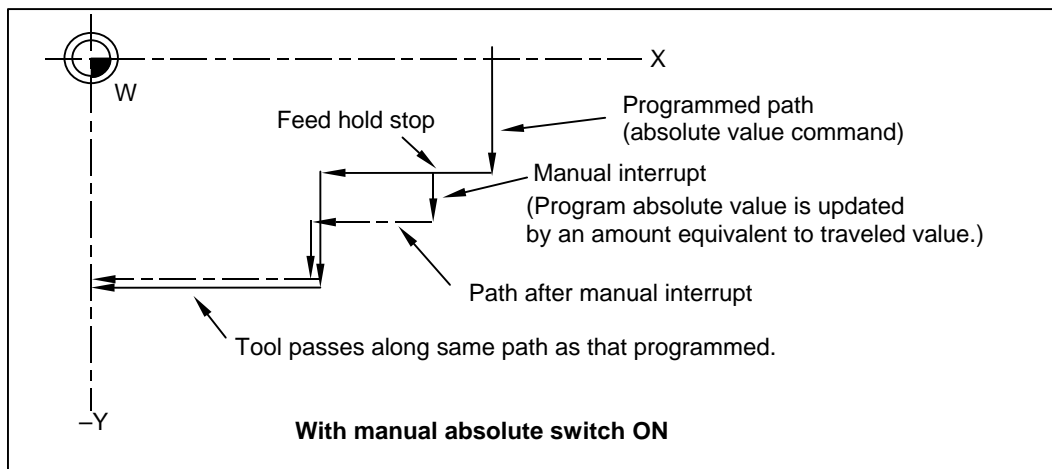
11 Operation Support Functions

11.4 Interrupt Operation

11.4.3 Manual Absolute Mode ON / OFF

M/L system	Model	M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system		○	○	○	○	○	○	○	○
L system		○	○	○	○	○	○	○	○

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



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

11 Operation Support Functions

11.4 Interrupt Operation

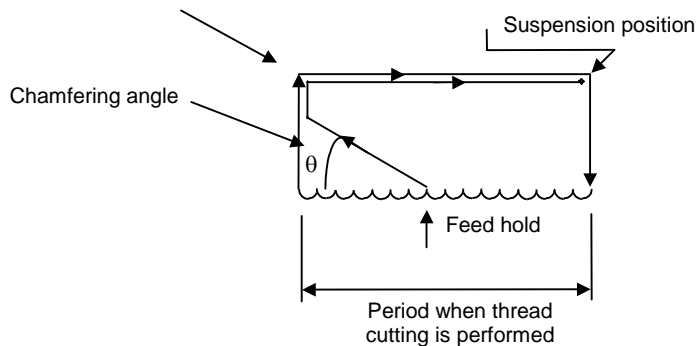
11.4.4 Thread Cutting Cycle Retract

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	-	-	-
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ

This function suspends the thread cutting cycle if a feed hold signal has been input during thread cutting in a thread cutting cycle.

If a feed hold signal is input during chamfering or thread cutting without chamfering, operation stops at the position where the block following the thread cutting is completed.

Position where the block following the thread cutting is completed



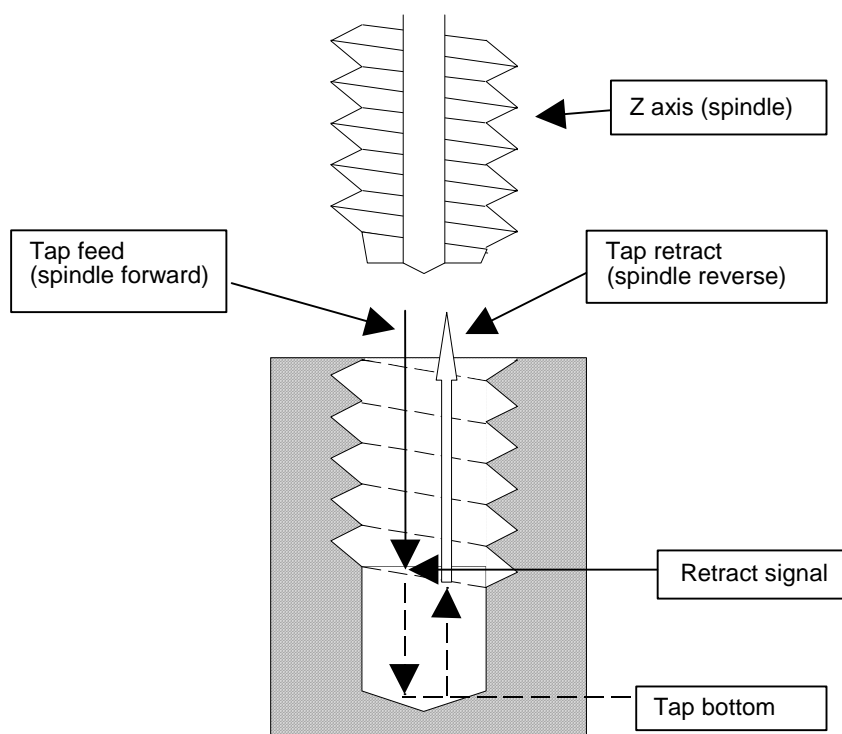
11 Operation Support Functions

11.4 Interrupt Operation

11.4.5 Tapping Retract

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

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 Operation Support Functions

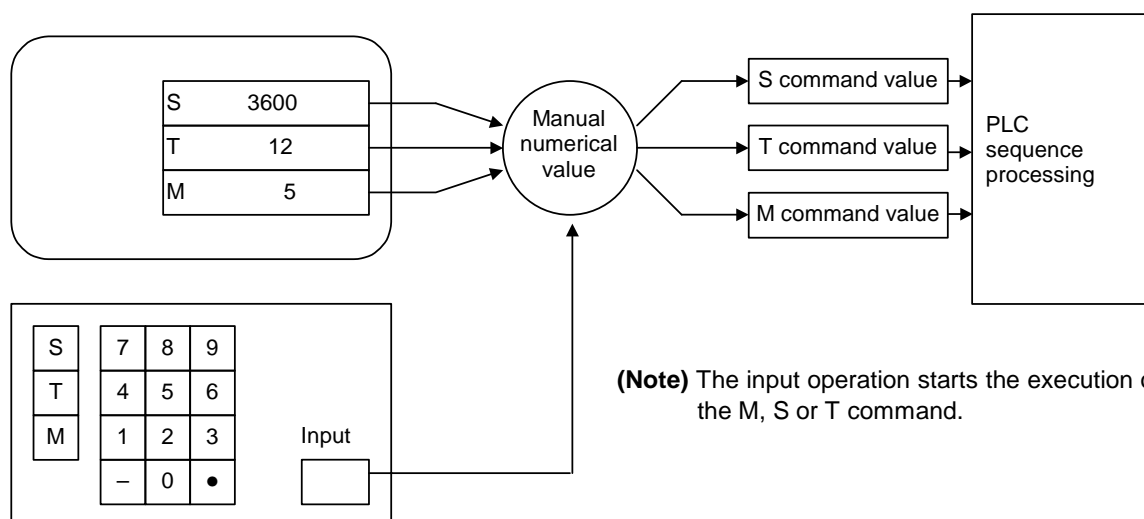
11.4 Interrupt Operation

11.4.6 Manual Numerical Value Command

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

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.



11.4.8 MDI Interruption

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

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.

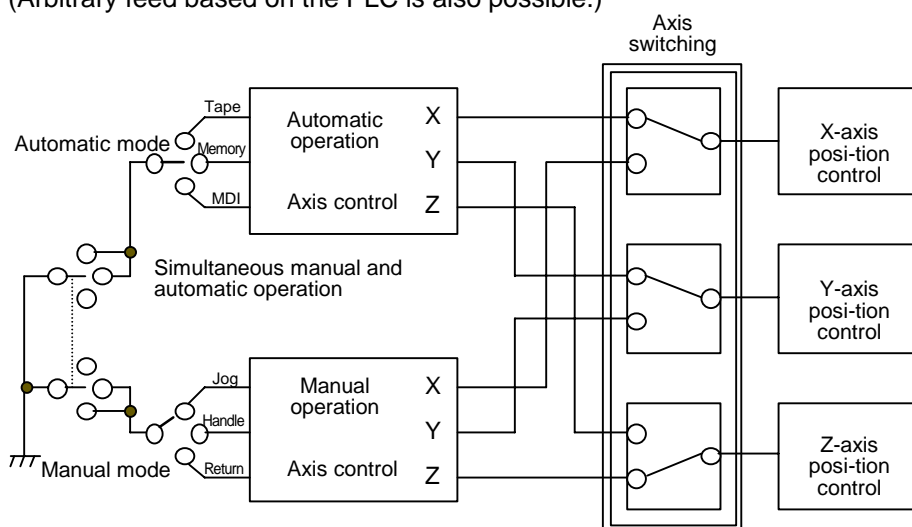
11 Operation Support Functions

11.4 Interrupt Operation

11.4.9 Simultaneous Operation of Manual and Automatic Modes

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

This function enables manual operations to be performed during automatic operation by selecting an automatic operation mode (tape, MDI or memory) and manual mode (handle, step, jog or manual reference position return) simultaneously.
(Arbitrary feed based on the PLC is also possible.)



The feed rates for the axes subject to automatic commands and the feed rates for axes subject to manual command are set separately. The acceleration/deceleration modes (rapid traverse, cutting feed) are also set separately. Rapid traverse override, cutting feed override and second cutting feed override are valid both for axes subject to automatic commands and 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 applied to axes subject to automatic commands.

11.4.10 Simultaneous Operation of JOG and Handle Modes

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

When executing the jog feed and handle feed, both these feeds are available without changing the mode each time by inputting the jog mode signal and simultaneous operation of jog and handle modes signal to the control unit. However, during moving in one of the two modes, the feed in the other mode is not valid.

11 Operation Support Functions

11.4 Interrupt Operation

11.4.11 Reference Position Retract

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		○	○	○	○	○	○	○	○
M system		○	○	○	○	○	○	○	○
L system		○	○	○	○	○	○	○	○

When the retract signal is turned ON during the automatic and manual operation, this function can retract the tool immediately to a set reference position.

The reference position to be retracted to can be selected from the 1st reference position to 4th reference position with 2-bit input signal.

Set the retracting order of axes with parameter (#2019 revnum).

(1) Other operations

- (a) When the retract signal is turned ON, the control unit is reset, the operation is interrupted, and the machining program is indexed.
- (b) When the rapid traverse input signal is input, the rapid traverse rate is applied. When the rapid traverse input signal is not input, the manual feed rate is applied.
- (c) If the retract signal is input during execution of a tapping cycle, the operation will be the tapping retract, and the normal reference position retract will be executed from the end point of tapping retract operation.
- (d) Even if the retract signal is input during the thread cutting cycle, it will be invalid. However, if the retract signal is input in a block other than the thread cutting block, the retracting operation will be executed.
- (e) If the retract signal is turned OFF midway during retracting, the operation will decelerate and stop. However, since the machining program is indexed, the block can not be resumed.
- (f) The retract signal is invalid if the coordinate system is not established. An operation error will occur when the retract signal is input in such case.

11.4.13 Skip Retract

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		-	-	-	-	-	Δ	Δ	Δ
M system		-	-	-	-	-	Δ	Δ	Δ
L system		-	-	-	-	-	-	-	-

This function is used to return to the opposite advance direction when the skip signal is input during G31 command.

11.4.14 PLC Interruption

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		-	-	-	-	Δ	Δ	Δ	Δ
M system		-	-	-	-	Δ	Δ	Δ	Δ
L system		-	-	-	-	Δ	Δ	Δ	Δ

The interrupt program set with the R register is executed with the signals from the PLC during single block stop in program operation or during the manual mode.

12 Programming Support Functions

12.1 Machining Method Support Functions

12 Program Support Functions

12.1 Machining Method Support Functions

12.1.1 Program

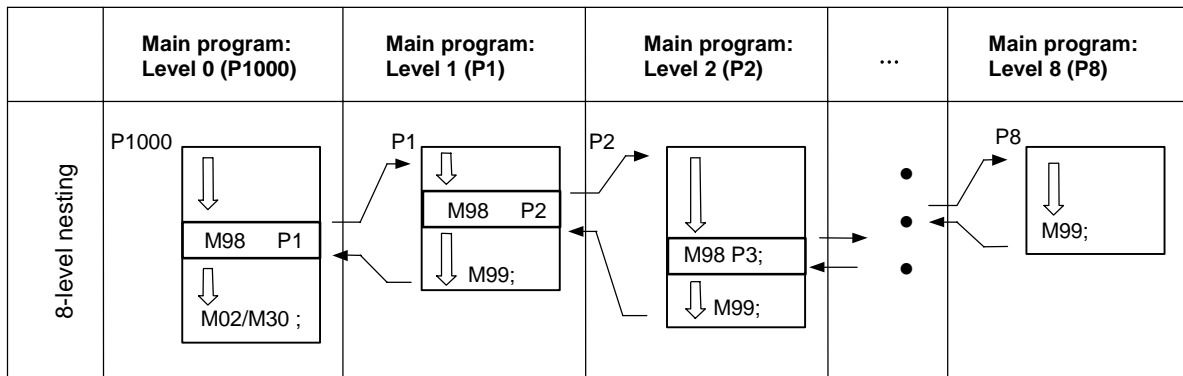
12.1.1.1 Subprogram Control

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○ 8 layers	○ 8 layers	○ 8 layers	○ 8 layers	○ 8 layers	○ 8 layers	○ 8 layers	○ 8 layers
L system	○ 8 layers	○ 8 layers	○ 8 layers	○ 8 layers	○ 8 layers	○ 8 layers	○ 8 layers	○ 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 can be made. The call is designated with the program number and sequence number.

<p>M98 Pp1 Hh1 LI1 ;</p> <p>M98 : Call command Pp1 : Subprogram number Hh1 : Sequence number LI1 : Number of repetitions</p> <p style="padding-left: 40px;">(Branch to subprogram) Op1 (Subprogram) : Nh1 : M99 ; (Return to main program)</p>

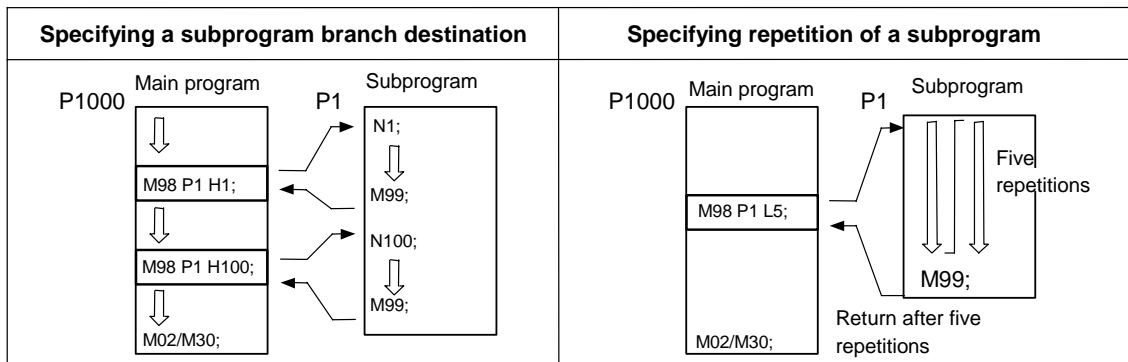
Subprograms can be nested up to eight levels deep.



12 Programming Support Functions

12.1 Machining Method Support Functions

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



12 Programming Support Functions

12.1 Machining Method Support Functions

12.1.1.3 Scaling

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	Δ	Δ	Δ	Δ	Δ	Δ
L system	-	-	-	-	-	-	-	-

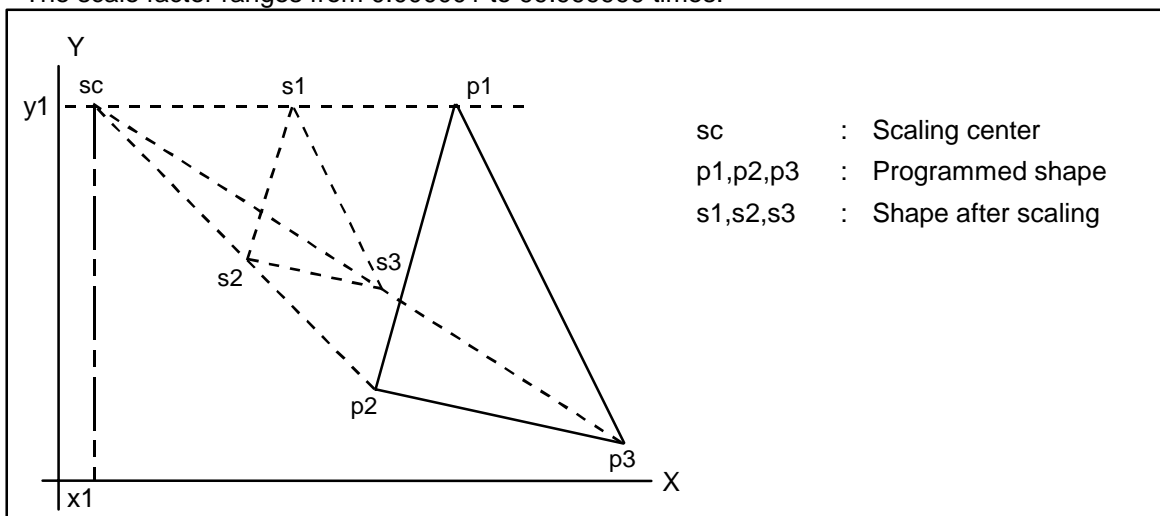
The shape commanded by the program can be extended or reduced to the desired size by applying a scale factor to the movement axis command value within the range assigned by the scaling command.

G code	Function
G50	Scaling cancel
G51	Scaling ON

The program format is given below.

<p>G51 Xx1 Yy1 Zz1 Pp1 ;</p> <p>G51 : Call command Xx1, Yy1, Zz1 : Scaling center coordinates Pp1 : Scale factor</p>

The scale factor ranges from 0.000001 to 99.999999 times.



(Note 1) Scaling cannot be applied to tool radius compensation, tool position offset, tool length offset or to other such offset amounts.

(Note 2) Scaling applies only to those axes X, Y and Z which have been commanded; it does not apply to axes which have not been commanded.
When the scale factor is not assigned, the parameter setting applies instead.

12 Programming Support Functions

12.1 Machining Method Support Functions

12.1.2 Macro Program

12.1.2.1 User Macro

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ 4 layers	Δ 4 layers	Δ 4 layers	Δ 4 layers	Δ 4 layers	Δ 4 layers	Δ 4 layers	Δ 4 layers
L system	Δ 4 layers	Δ 4 layers	Δ 4 layers	Δ 4 layers	Δ 4 layers	Δ 4 layers	Δ 4 layers	Δ 4 layers

(1) Macro commands (1) ; G65 to G67

In order to carry through one integrated function, a group of control and arithmetic instructions can be used and registered as a macro program. Furthermore, subprograms with a high degree of expandability can be configured by setting these macro programs as types which are capable of conducting control and arithmetic operations using variable commands.

G code	Function
G65	Macro call (Sample call)
G66	Macro modal call A
G66.1	Macro modal call B
G67	Macro modal call cancel

The program formats are given below.

G65 Pp1 LI1 Argument ;
G65 : Call command
Pp1 : Program No.
LI1 : No. of repetitions
Argument : Variable data assignment

The macro program is called immediately by this command.

G66 Pp1 LI1 Argument ;
G66 : Call command
Pp1 : Program No.
LI1 : 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 LI1 Argument ;
G66.1 : Call command
Pp1 : Program No.
LI1 : No. of repetitions
Argument : Variable data assignment

The macro program is executed with the word data of each block as the argument.

12 Programming Support Functions

12.1 Machining Method Support Functions

The following macro command functions are available.

Arithmetic commands	#1 = <Expression> ; Various arithmetic operations can be conducted between variables by the above. "<Expression>" is a combination of constants, variables, functions and operators.
Assignment of priority of arithmetic operations	The portion in which the operator is to be given priority can be enclosed in []. Up to five pairs of square parentheses [] including the function [] can be used. The normal priority of operation is functions and multiplication/division followed by addition/subtraction.
Control commands	(1) IF [<Conditional expression>] GOTO n ; (2) WHILE [<Conditional expression>] DO m ; . . . END m ; The flow of the program can be controlled by these commands. "n" denotes the sequence numbers of the branching destination. "m" is an identification number, and 1 to 127 can be used. Note that only 27 nestings can be used.

(Note) The variable commands are provided under the optional specifications independently of the user macros. If they are to be used, specify the optional specifications separately.

(2) Macro commands (2)

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

(a) Macro call using G codes

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

Format

GXX	<Argument>	;	
GXX		:	G code for performing macro call

The correspondence between the G_{xx} 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. [M system].)
- Up to 800 codes from G200 to G999 can be used in this command. [L system]

12 Programming Support Functions

12.1 Machining Method Support Functions

(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.)

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]

12.1.2.2 Machine Tool Builder Macro

12.1.2.2.1 Machine Tool Builder Macro SRAM

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ

This function enables macro programs exclusively designed for use by the machine builders to be registered in addition to the regular macro programs. These macros can be called from user programs using the same method as the one used for regular macros. Machine builder macros can be locked, preventing them from being viewed unless the key word is input.

Machine builder macro programs are stored in a dedicated area which means that the user program registration area is not reduced in the process.

Call format 1

G65 Pp1 LI ;

P : Machine builder macro program number (0100001000 – 0199999998)

I : Number of repetitions

Note: Machine builder macros cannot be called using the G66, G66.1 or M98 command.

Call format 2

G* ;**

*** : G code defined in macro definition program

12 Programming Support Functions

12.1 Machining Method Support Functions

12.1.2.3 Macro Interruption

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	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.

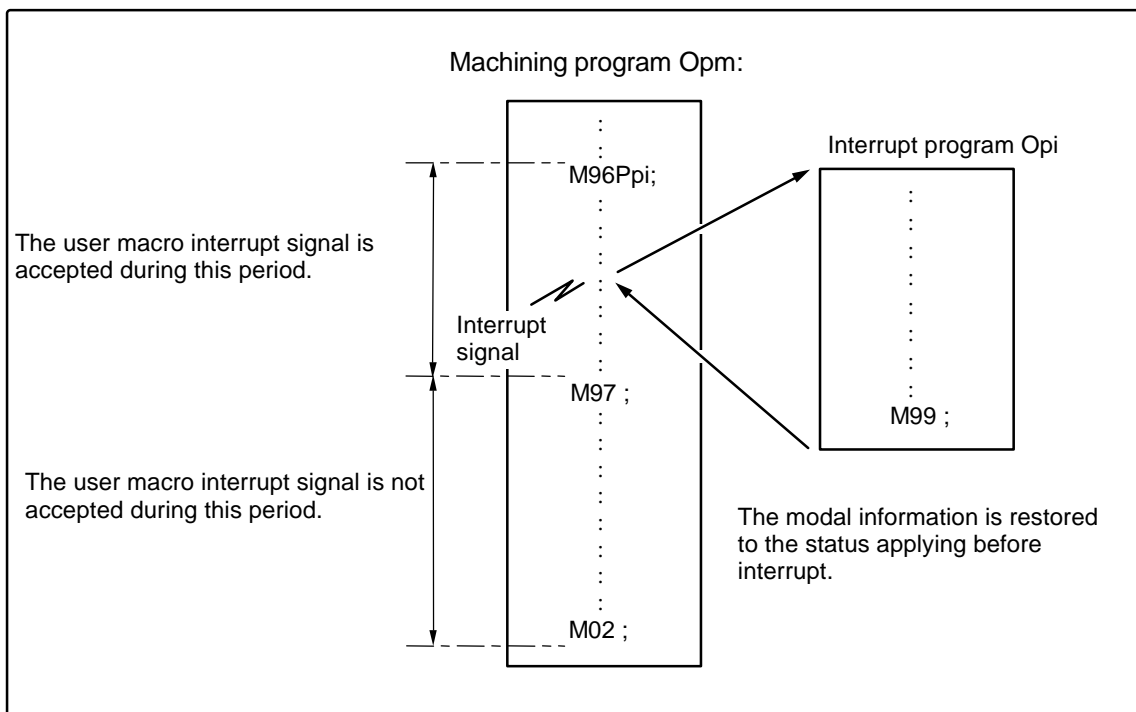
Retract or return operations when tools have been damaged, for instance, and other kinds of restoration operations to be conducted when trouble has occurred are programmed in the interrupt programs. There are two types of interrupts, type 1 and type 2, as described below, and they are selected using 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 Programming Support Functions

12.1 Machining Method Support Functions

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.

Number of variable sets specifications		
System 1	100 sets	#100 ~ #149, #500 ~ #549
	200 sets	#100 ~ #199, #500 ~ #599
	300 sets	#100 ~ #199, #500 ~ #699
	600 sets	#100 ~ #199, #500 ~ #999
	700 sets	#100 ~ #199, #400 ~ #999
System 2	50 + 50 sets	#100 ~ #149 (per part system), #500 ~ #549 (common to part system)
	100 + 100 sets	#100 ~ #199 (per part system), #500 ~ #599 (common to part system)
	100 + 400 sets	#100 ~ #199 (per part system), #500 ~ #899 (common to part system)
	100 + 500 sets	#100 ~ #199 (per part system), #500 ~ #999 (common to part system)

- Variable names can be set for #500 ~ #519.

Variable expressions

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

Variable definition

Variable = expression

(Note 1) Variables cannot be used with addresses "O" and "N".

12.1.2.4.1 100 sets

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

12.1.2.4.2 200 sets

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ

12 Programming Support Functions

12.1 Machining Method Support Functions

12.1.2.4.3 300 sets

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ

12.1.2.4.4 600 sets

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	-	-	-	Δ	Δ	Δ
L system	Δ	Δ	-	-	-	Δ	Δ	Δ

12.1.2.4.5 700 sets

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	-	-	-	-	-	-
L system	Δ	Δ	-	-	-	-	-	-

12.1.2.4.6 (50+50 x number of part systems) sets

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	○	○	○	-	-	-	-
L system	-	○	○	○	-	○	○	○

12.1.2.4.7 (100+100 x number of part systems) sets

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	Δ	Δ	Δ	-	-	-	-
L system	-	Δ	Δ	Δ	-	Δ	Δ	Δ

12.1.2.4.8 (200+100 x number of part systems) sets

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	Δ	Δ	Δ	-	-	-	-
L system	-	Δ	Δ	Δ	-	Δ	Δ	Δ

12.1.2.4.9 (400+100 x number of part systems) sets

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	-	-	-
L system	-	-	-	-	-	Δ	Δ	Δ

12.1.2.4.10 (500+100 x number of part systems) sets

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	Δ	-	-	-	-	-	-
L system	-	Δ	-	-	-	-	-	-

12 Programming Support Functions

12.1 Machining Method Support Functions

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	G code system 6	G code system 7		
Fixed cycle for drilling	G70	G80	G80	G80	G80	Refer to 12.1.3.1. Refer to 4.5.3.	
	:	:	:	:	:		
	G89	G89	G89	G89	G89		
	G98	G79	G83.2	G79	G83.2		
	G99	G98	G98	G98	G98		
	G99	G99	G99	G99	G99		
Special fixed cycles	G34 G35 G36	—	—	—	—	Refer to 12.1.3.2.	
Fixed cycles for turning machining	—	G90 G92 G94	G77 G78 G79	G90 G92 G94	G77 G78 G79	Refer to 12.1.3.3.	
Multiple repetitive fixed cycles for turning machining	—	G70 : G76 G76.1 G76.2	G70 : G76 G76.1 G76.2	G70 : G76 G76.1 G76.2	G70 : G76 G76.1 G76.2	Refer to 12.1.3.4. Refer to 12.1.3.5. Refer to 4.5.8.	
Small-diameter deep-hole drilling cycle	G83	—	—	—	—	Refer to 12.1.3.6.	

12 Programming Support Functions

12.1 Machining Method Support Functions

12.1.3.1 Fixed Cycle for Drilling

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

(1) M series ; 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

12 Programming Support Functions

12.1 Machining Method Support Functions

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

G81	Xx1	Yy1	Zz1	Rr1	Qq1	Pp1	Ll1	Ff1	;
G81									
Xx1, Yy1									
Zz1									
Rr1									
Qq1									
Pp1									
Ll1									
Ff1									

For details on the synchronous tapping cycle (including pecking tapping cycle and deep-hole tapping cycle), refer to the section "4.5.3 Synchronous Tapping".

12 Programming Support Functions

12.1 Machining Method Support Functions

<p style="text-align: center;">G73 Step cycle</p>	<p style="text-align: center;">G74 Reverse tapping cycle</p>	<p style="text-align: center;">G76 Fine boring cycle</p>	<p style="text-align: center;">G81 Drilling, spot drilling cycle</p>
<p style="text-align: center;">G82 Drilling, counterboring cycle</p>	<p style="text-align: center;">G83 Deep hole drilling cycle</p>	<p style="text-align: center;">G84 Tapping cycle</p>	<p style="text-align: center;">G85 Boring cycle</p>
<p style="text-align: center;">G86 Boring cycle</p>	<p style="text-align: center;">G87 Back boring cycle</p>	<p style="text-align: center;">G88 Boring cycle</p>	<p style="text-align: center;">G89 Boring cycle</p>

12 Programming Support Functions

12.1 Machining Method Support Functions

(2) L series ; 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

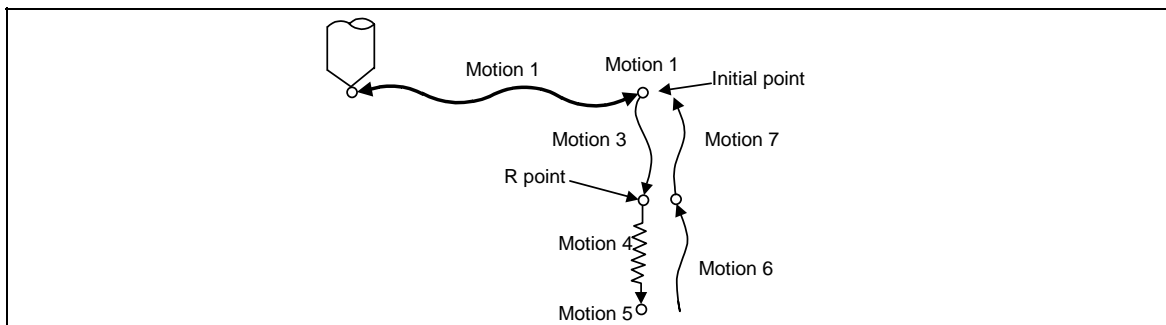
G83/G84/G85	Xx1 Cc1 Zz1 Rr1 Qq11 Pp1 Ff1 Kk1 (Mm1) Ss1 ,Ss1 Dd1 ,Rr1 ;
G87/G88/G89	Xx1 Cc1 Zz1 Rr1 Qq11 Pp1 Ff1 Kk1 (Mm1) Ss1 ,Ss1 Dd1 ,Rr1 ;
G83/G84/G85	: Fixed cycle mode of drilling (G83, G87), tapping (G84, G88), or boring (G85, G89)
G87/G88/G89	The drilling command is modal. Once it is given, it 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)

12 Programming Support Functions

12.1 Machining Method Support Functions

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 of return
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: 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:

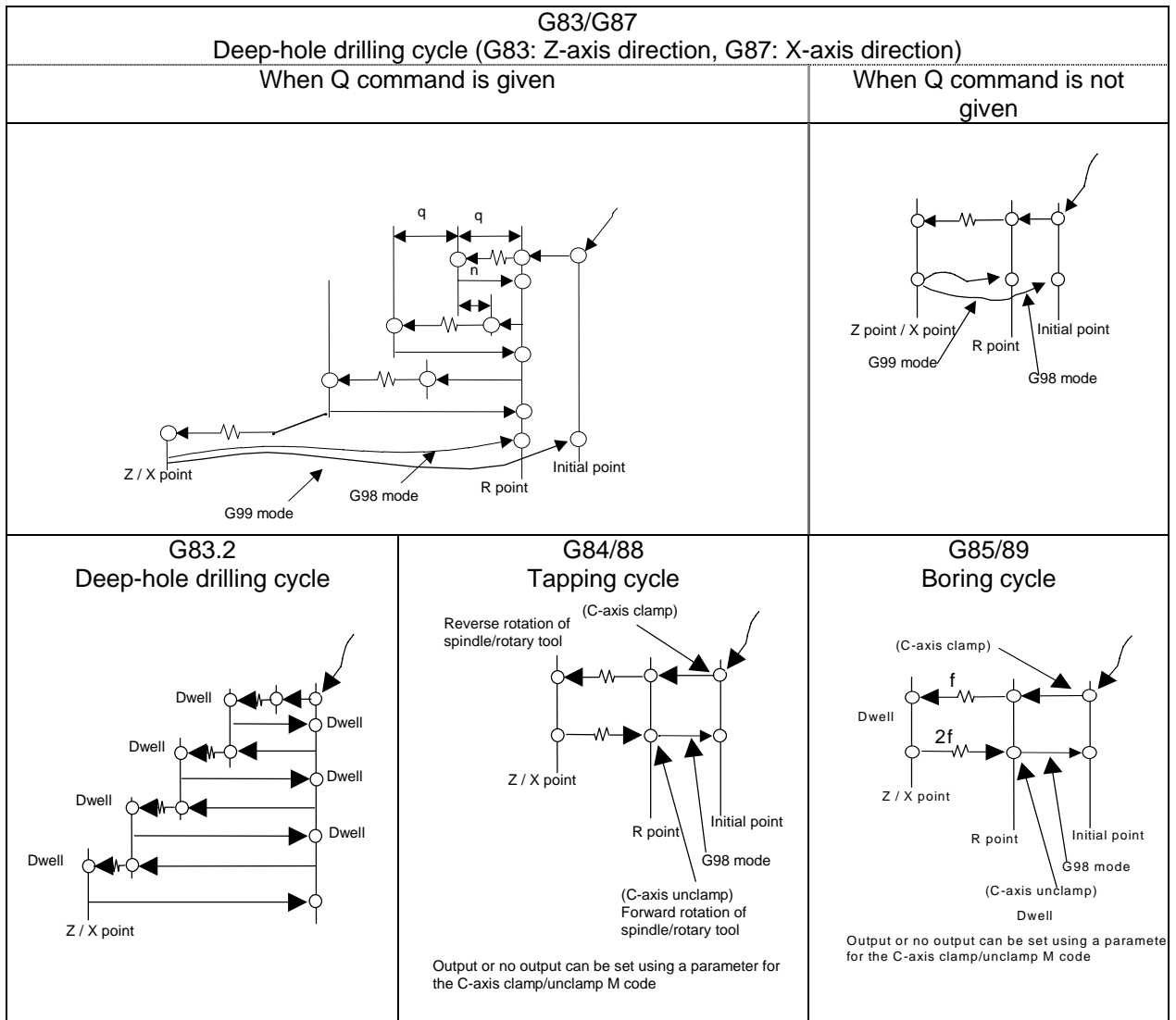
- G98: Initial level return
- G99: R point level return

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.

12 Programming Support Functions

12.1 Machining Method Support Functions

Deep-hole drilling cycle (G83, G87)



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

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

12 Programming Support Functions

12.1 Machining Method Support Functions

12.1.3.2 Special Fixed Cycle

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	-	-	-	-	-	-	-	-

Special fixed cycles must always be used in combination with fixed cycles.

(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 ;
  
```

Position prior to execution of G34 command (500 mm, 100 mm) G0 command in N005

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.

12 Programming Support Functions

12.1 Machining Method Support Functions

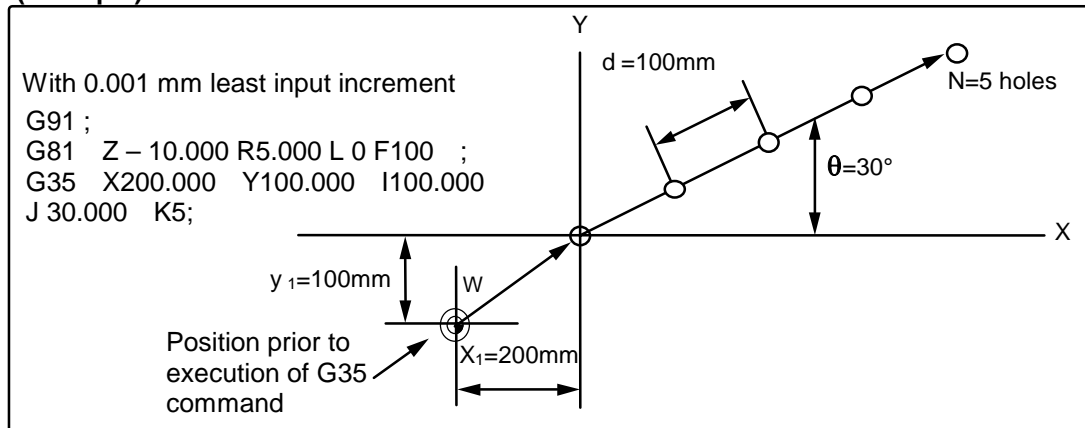
(2) Line at angle (G35)

With the starting point at the position designated by X and Y, the tool drills "n" number of holes each at interval "d" in the direction forming angle θ with the X axis. A standard fixed cycle applies for the drilling operation at each of the hole positions and so there is a need to retain beforehand the drilling data (drilling mode and drilling data). All movements between the hole positions are conducted in the G00 mode. The data is not retained upon completion of the G35 command.

G35 Xx Yy Id J θ Kn ;

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

(Example)



12 Programming Support Functions

12.1 Machining Method Support Functions

(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 J θ P $\Delta\theta$ Kn ;

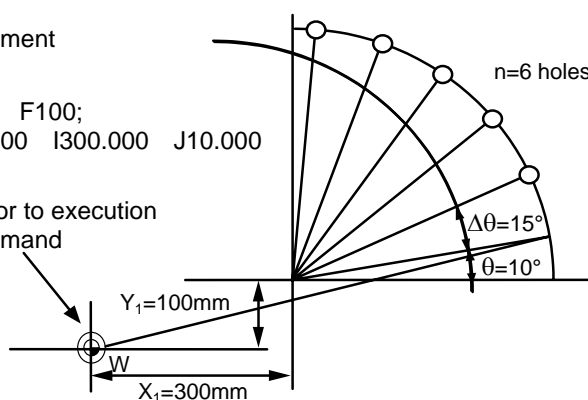
Xx, Yy : Center coordinates of arc; they are affected by the G90/G91 commands.
 Ir : Radius "r" of arc; it is based on the least input increment and is provided with a positive number.
 J θ : Angle θ at the point to be drilled initially; the counterclockwise direction is taken to be positive.
 P $\Delta\theta$: Angle interval $\Delta\theta$; when it is positive, the tool drills in the counterclockwise direction and when it is negative, it drills in the clockwise direction.
 Kn : Number "n" of holes to be drilled; any number of holes from 1 through 9999 can be assigned.

(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;
```

Position prior to execution
of G36 command



12 Programming Support Functions

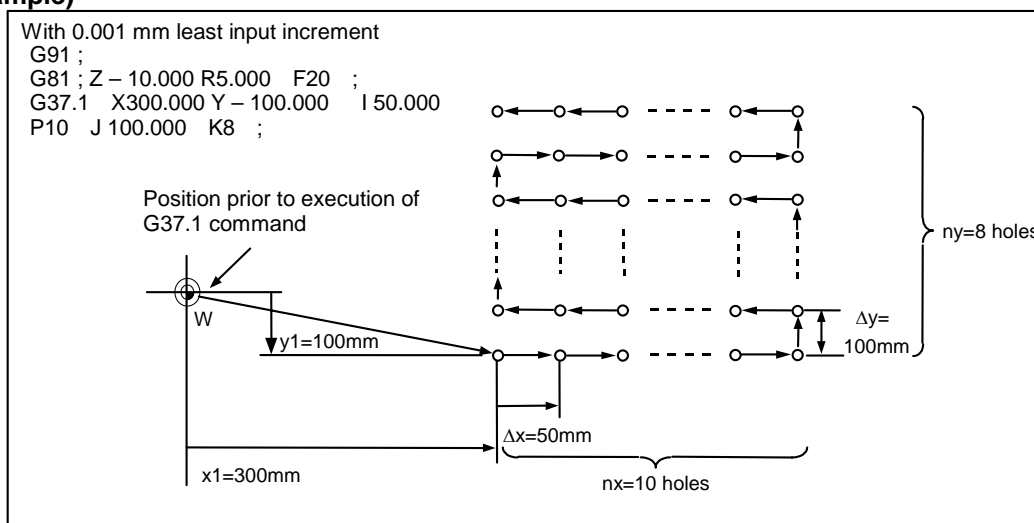
12.1 Machining Method Support Functions

(4) Grid (G37.1)

With the starting point at on the position designated by X and Y, this function enables the tool to drill the holes on the lattice with "nx" number of holes at parallel intervals of Δx to the X axis. Drilling proceeds in the X-axis direction. The drilling operation at each of the hole positions is based on a standard fixed cycle and so there is a need to command the drilling data (drilling mode and drilling data) beforehand. All movements between the hole positions are conducted in the G00 mode. The data is not retained upon completion of the G37.1 command.

G37.1 Xx1 Yy1 IΔx Pnx JΔy Kny ;	
Xx, Yy	: The starting point coordinates; they are affected by the G90/G91 commands.
IΔx	: X-axis interval Δx ; it is based on the least input increment; when Δx is positive, the intervals are provided in the positive direction as seen from the starting point and when it is negative, they are provided in the negative direction.
Pnx	: Number of holes "nx" in the X-axis direction; any number of holes from 1 through 9999 can be assigned.
JΔy	: Y-axis interval Δy ; it is based on the least input increment; when Δy is positive, the intervals are provided in the positive direction as seen from the starting point and when it is negative, they are provided in the negative direction.
Kny	: Number of holes "ny" in the Y-axis direction; any number of holes from 1 through 9999 can be assigned.

(Example)



12 Programming Support Functions

12.1 Machining Method Support Functions

12.1.3.3 Fixed Cycle for Turning Machining

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	-	-	-
L system	○	○	○	○	○	○	○	○

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:

GΔΔ 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

12 Programming Support Functions

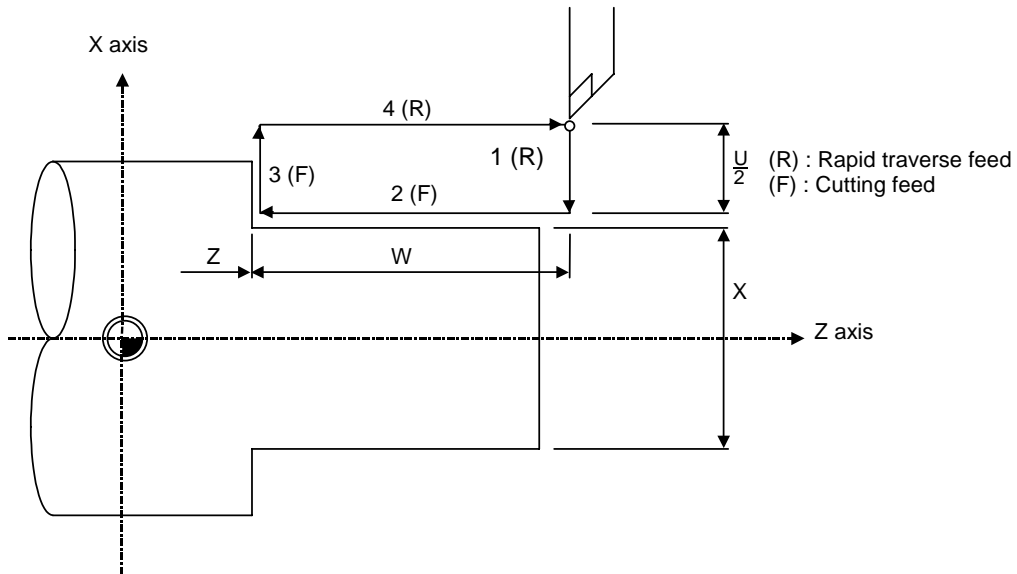
12.1 Machining Method Support Functions

(1) Longitudinal cutting cycle (G77)

(a) Longitudinal cutting

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

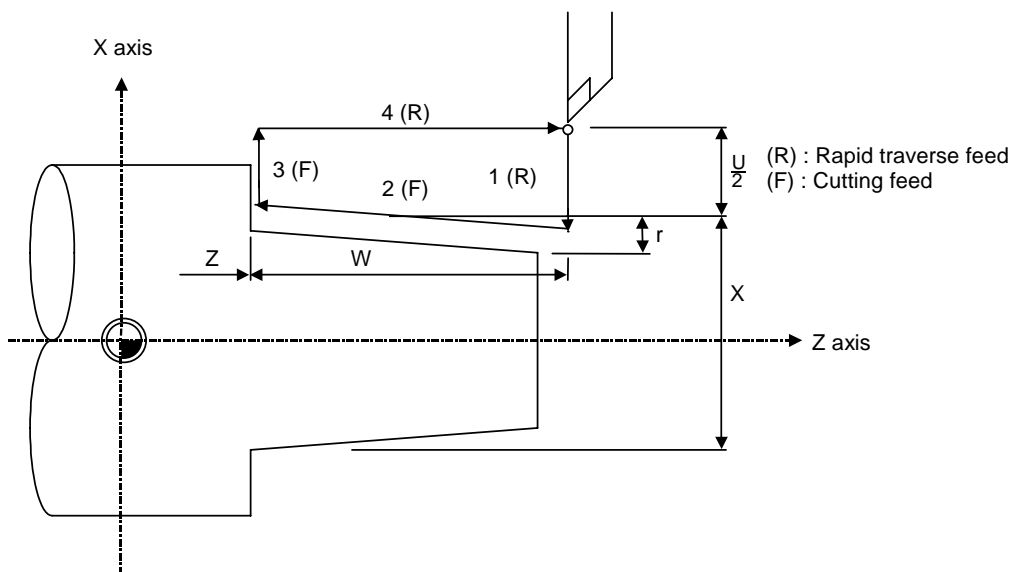
G77 X/U_ Z/W_ F_ ;



(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)

12 Programming Support Functions

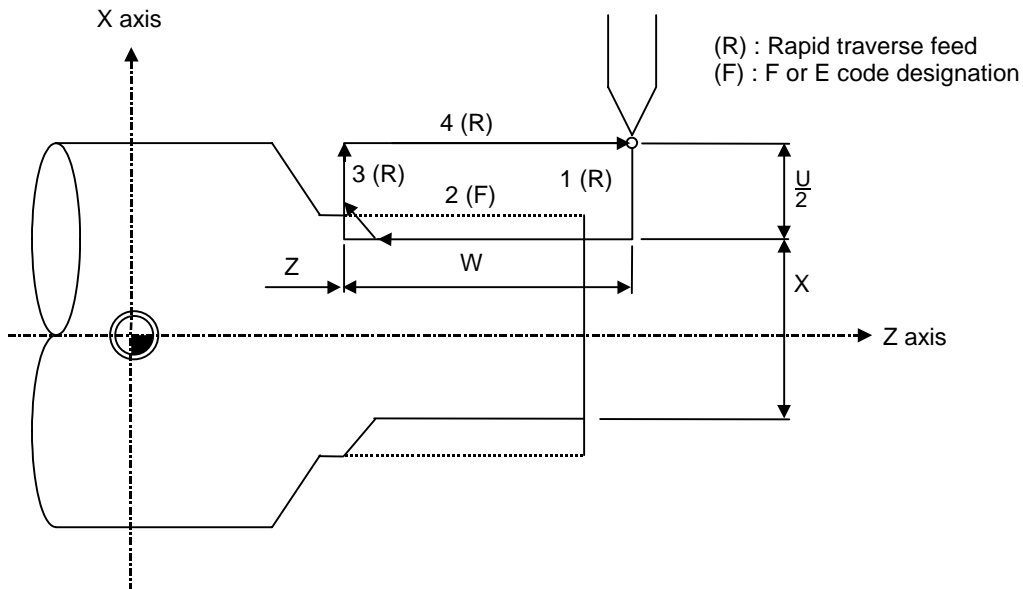
12.1 Machining Method Support Functions

(2) Thread cutting cycle (G78)

(a) Straight thread cutting

Straight thread cutting can be performed by the following block:

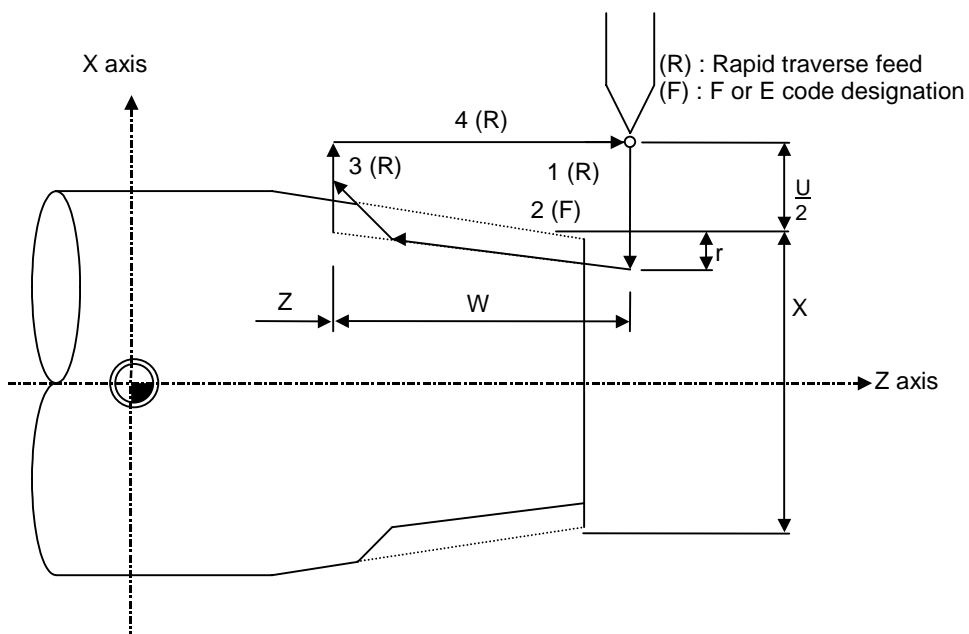
G78 X/U_ Z/W_ F/E_ ;



(b) Taper thread cutting

Taper thread cutting can be performed by the following block:

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

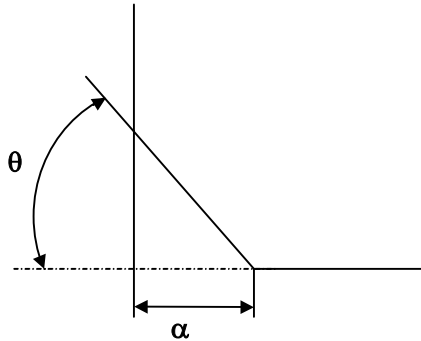


r: Taper part depth (radius designation, incremental value, sign is required)

12 Programming Support Functions

12.1 Machining Method Support Functions

Chamfering



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

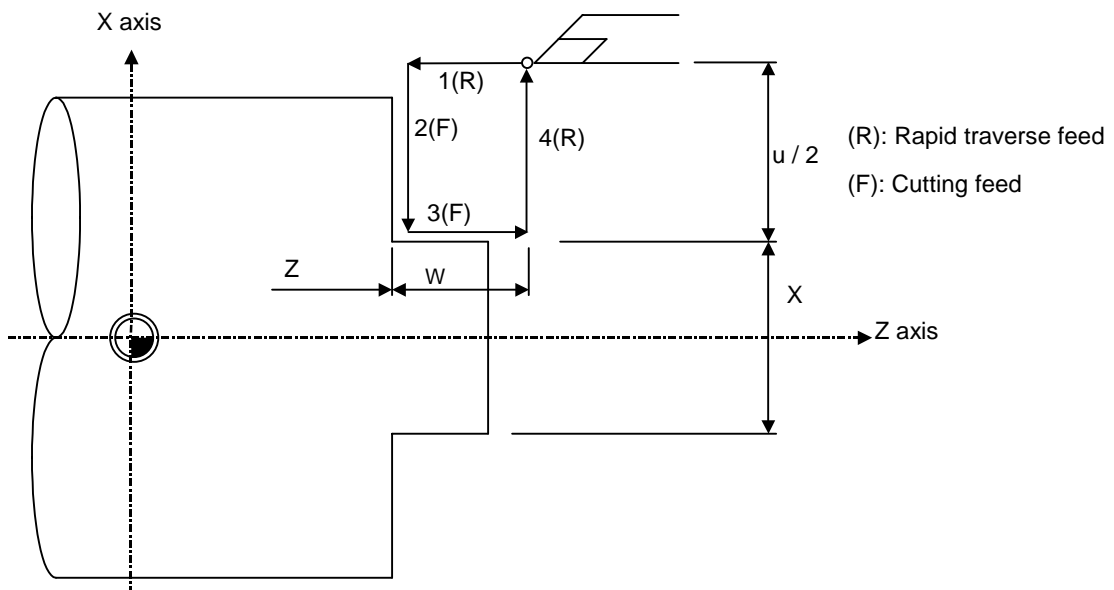
θ : Thread cutting-up angle
The thread cutting-up angle can be set in a given parameter in 1° steps in the range of 0 to 89°.

(3) Face cutting cycle (G79)

(a) Straight cutting

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

G79 X/U_ Z/W_ F_ ;



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

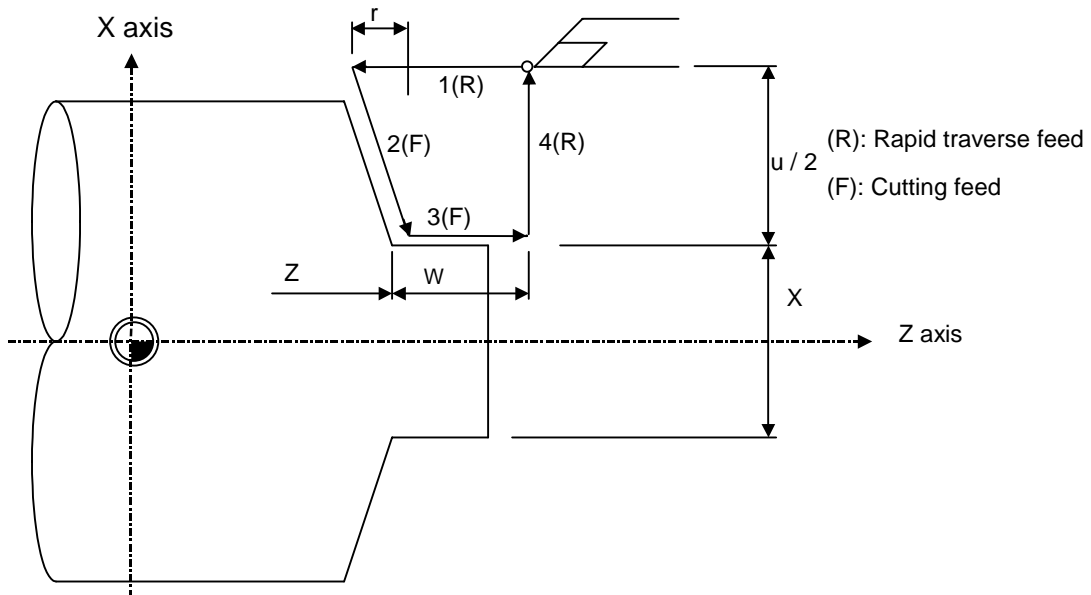
12 Programming Support Functions

12.1 Machining Method Support Functions

(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 Programming Support Functions

12.1 Machining Method Support Functions

12.1.3.4 Multiple Repetitive Fixed Cycle for Turning Machining

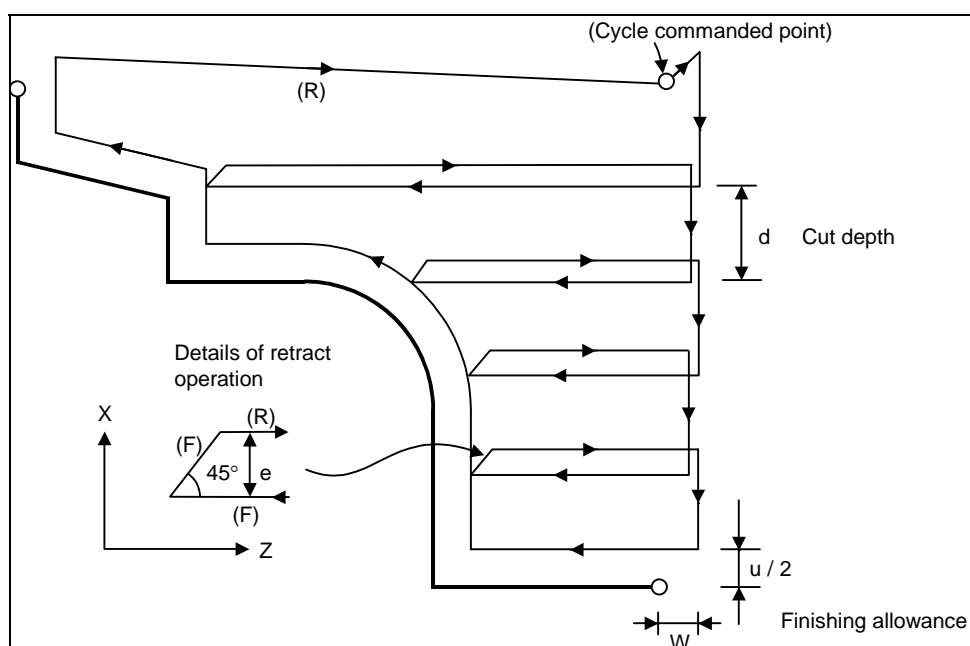
M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	-	-	-
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ

(a) Longitudinal rough cutting cycle I (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.	



12 Programming Support Functions

12.1 Machining Method Support Functions

(b) Face rough cutting cycle (G72)

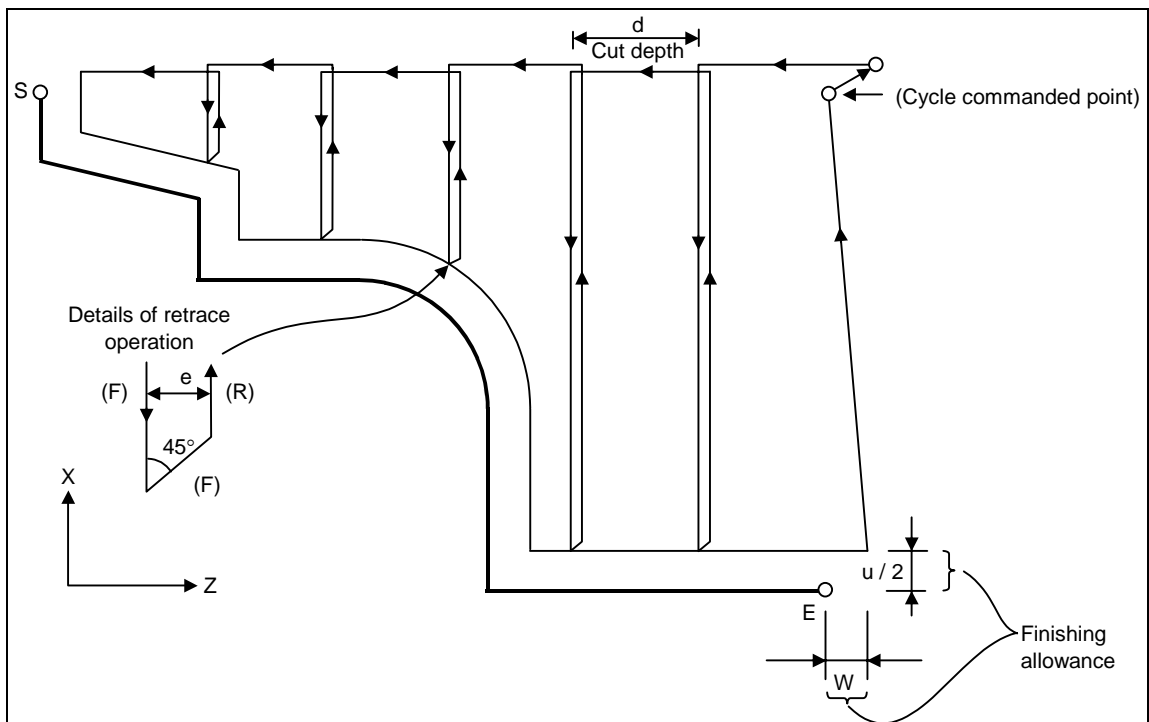
The finish shape program is called, and rough turning is performed in the end face direction while intermediate path is being calculated automatically.

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.	



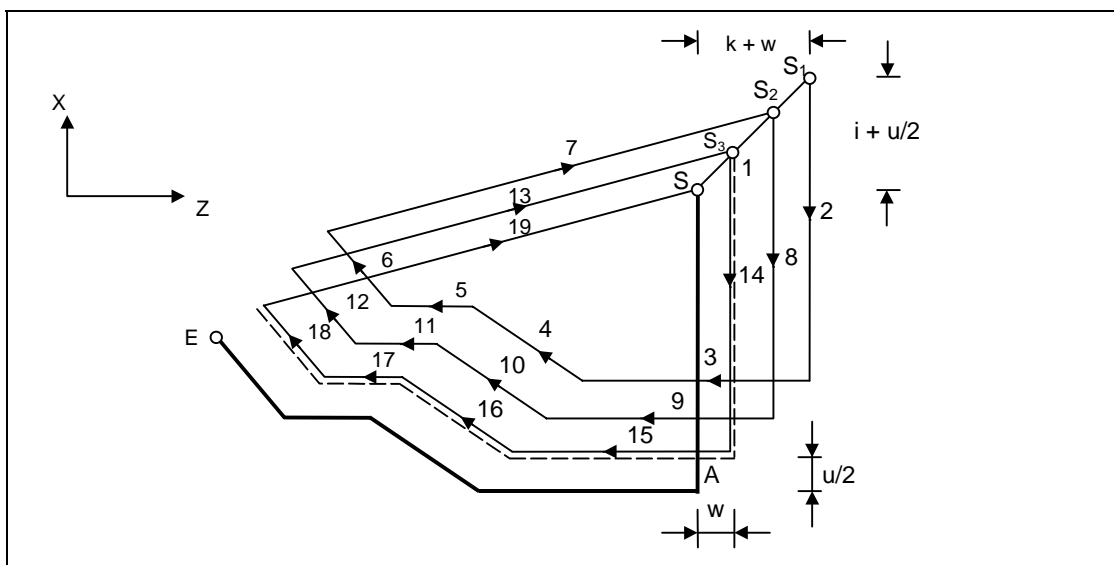
12 Programming Support Functions

12.1 Machining Method Support Functions

(c) 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 Wk : Cutting allowance in the Z axis direction Rd : Split count	i k d	} <ul style="list-style-type: none"> • Cutting allowance when P, Q command is not given. • Modal data • Sign is ignored. • Cutting allowance is given with a radius designation.
Aa : Finish shape program No. Pp : Finish shape start sequence No. Qq : Finish shape end sequence No.		(If it is omitted, the present program is assumed to be designated.) (If it is omitted, the program top is assumed to be designated.) (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 Ww : Finishing allowance in the Z axis direction	u w	} <ul style="list-style-type: none"> • 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.
Ff : Cutting feed rate (F function) Ss : Spindle speed (S function) Tt : Tool selection (T 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.



12 Programming Support Functions

12.1 Machining Method Support Functions

(d) Finish cycle (G70)

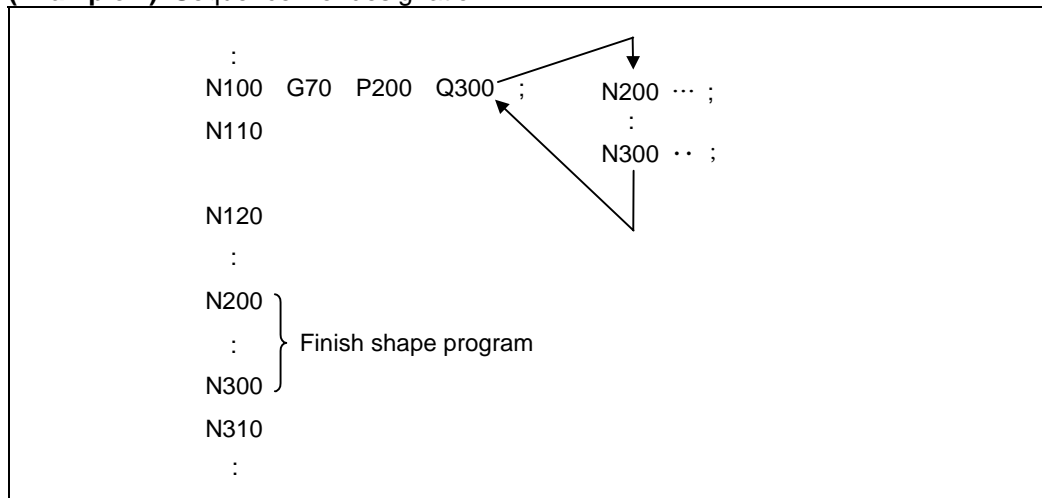
After rough cutting is performed by using G71 to G73, finish turning can be performed by using the G70 command.

The machining program is commanded as follows.

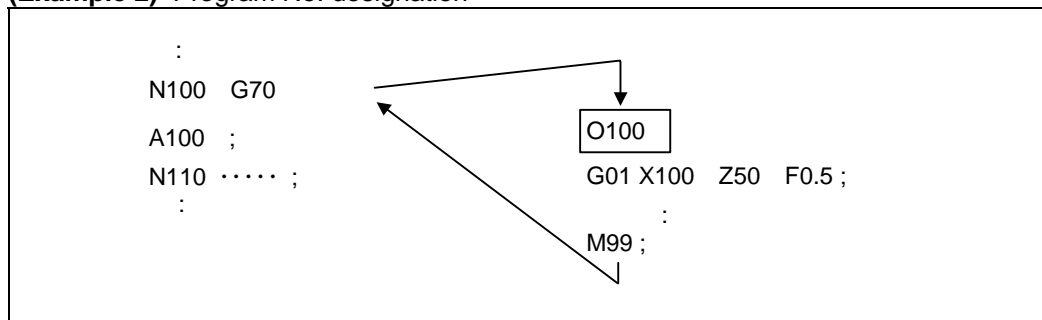
G70 A_P_Q_;	
A	: Finish shape program number. (If it is omitted, the program being executed is assumed to be designated.)
P	: Finish shape start sequence number. (If it is omitted, the program top is assumed to be designated.)
Q	: Finish shape end sequence number. (If it is omitted, the program end is assumed to be designated.)
However, if M99 precedes the Q command, up to M99.	

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

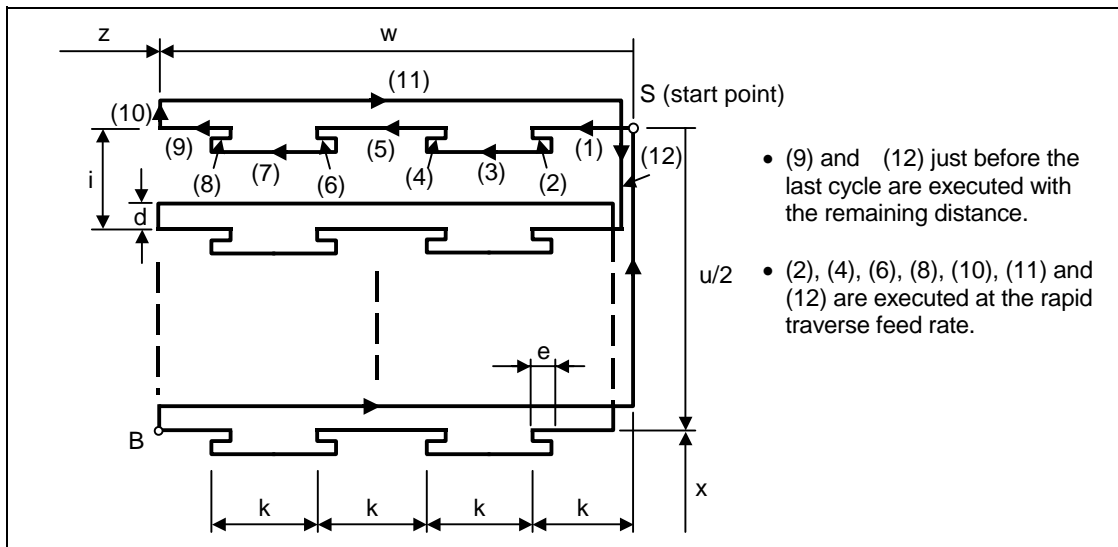
12 Programming Support Functions

12.1 Machining Method Support Functions

(e) 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



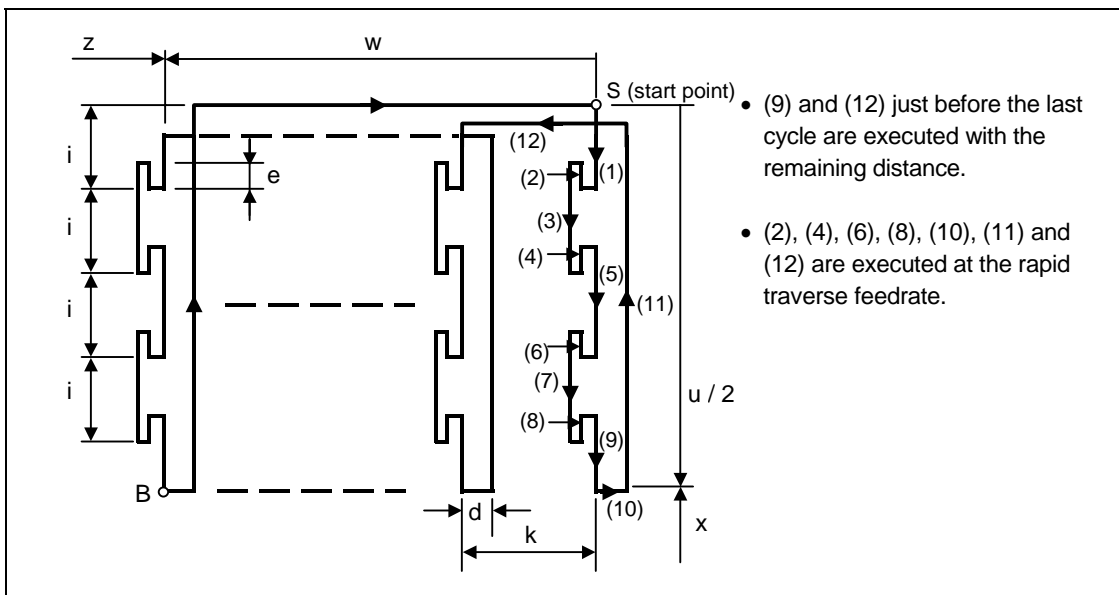
12 Programming Support Functions

12.1 Machining Method Support Functions

(f) 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)
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 ⊖ sign is provided, relief is made not at the first cut bottom but at the second cut bottom and later.)
Ff	: Feed rate



12 Programming Support Functions

12.1 Machining Method Support Functions

(g) 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

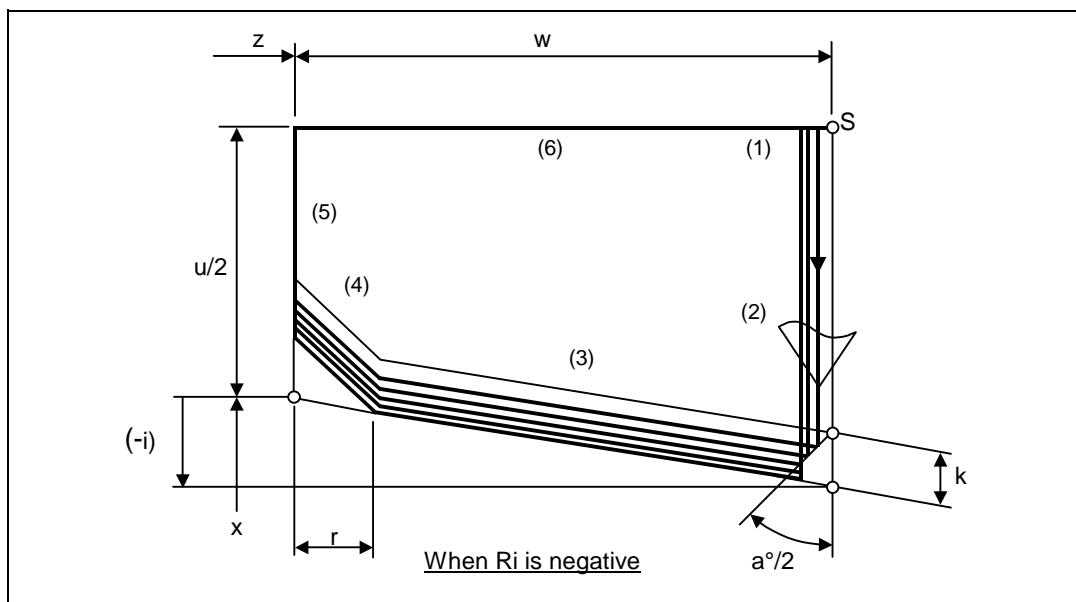
G76 Pmra Rd ;

G76 X/U Z/W Ri Pk QΔd FI ;

m	:	Cut count at finishing 01 to 99 (modal)
r	:	Chamfering amount 00 to 99 (modal). Set in 0.1-lead increments.
a	:	Nose angle (included angle of thread) 00 to 99 (modal) Set in 1-degree increments.
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

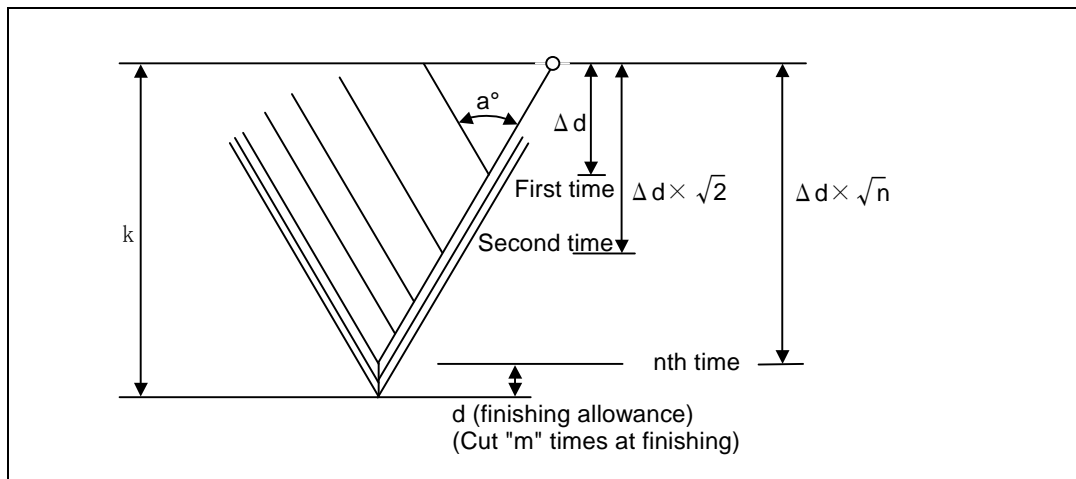
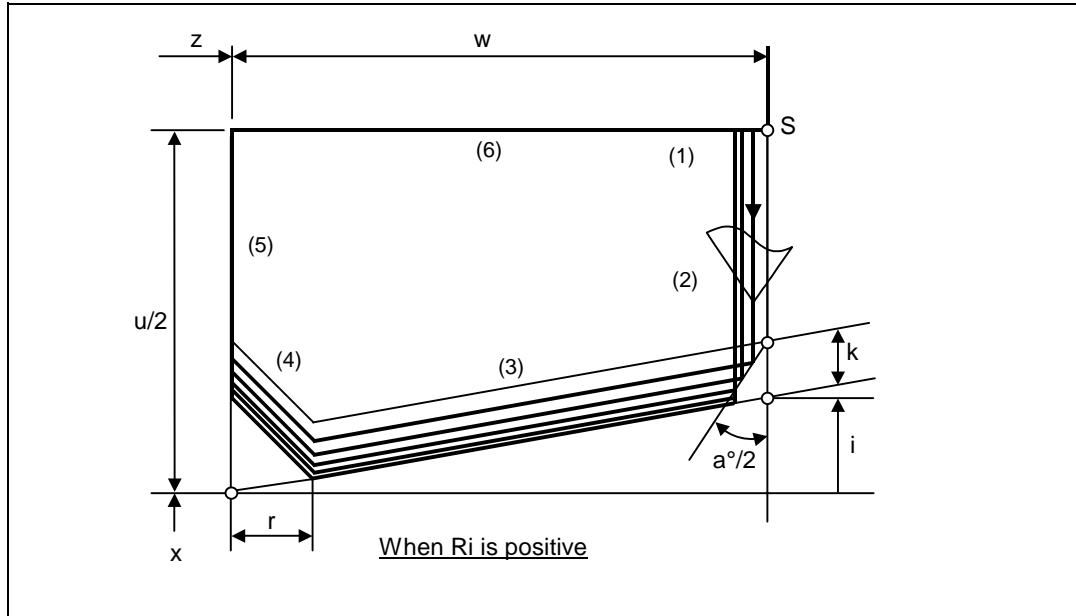
Configuration of one cycle

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 Programming Support Functions

12.1 Machining Method Support Functions



12 Programming Support Functions

12.1 Machining Method Support Functions

12.1.3.5 Multiple Repetitive Fixed Cycle for Turning Machining (Type II)

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	-	-	-
L system	-	-	-	-	-	Δ	Δ	Δ

Pocket shapes can be machined in the longitudinal rough cutting cycle (G71) and face rough cutting cycle (G72).

Command format (This is a command format when the G71 is commanded. The G72 command is based on the G71 command.)

G71 Ud Re Hh ; ← (can be omitted when values set in parameters are used) G71 Aa Pp Qq Uu Ww Ff Ss Tt ;	
<H0: Used for finished shapes without pockets>	<H1: Mainly used for finished shapes with pockets>
<p>G71 Ud Re H0; G71 Pp Qq · · ;</p>	<p>G71 Ud Re H1; G71 Pp Qq · · ;</p>
<p>Hh : Pocket machining (modal) ... reversible parameter ("G71 pocket machining")</p> <p>0: Select this only for finished shapes without hollow areas (pockets).</p> <p>* With the beginning of the pockets, the tool is pulled up in the 45-degree direction with each cycle until the finished shape is finally traced.</p> <p>1: This can be selected regardless of whether the finished shape has hollow (pocket) parts or not.</p> <p>* A method that traces the finished shape with each cycle is used for the beginning of the pockets.</p>	

12 Programming Support Functions

12.1 Machining Method Support Functions

12.1.3.6 Small-diameter Deep-hole Drilling Cycle

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	Δ	Δ	Δ
L system	-	-	-	-	-	-	-	-

In deep hole drilling, cutting and retract are repeated and the workpiece is machined multiple times. In addition, when PLC signals are input during cutting, the cutting for the time concerned is skipped. In this way, this cycle reduces the load applied to the tool.

Command format

The small-diameter deep hole drilling cycle mode is established by designating the M code command which was set in the parameter. If the G83 command is designated in this mode, the small-diameter deep hole drilling cycle is executed. The mode is canceled by the following conditions.

- Designation of a fixed cycle cancel command (G80, G commands in Group 1)
- Resetting

The small-diameter deep hole drilling mode M command does not output MF or wait for FIN.

G83 Xx1 Yy1 Zz1 Rr1 Qq1 Ff1 Pp1 ;	
X, Y	: Hole drilling point position
Z	: Hole bottom position
R	: R point position
Q	: Depth of cut in each pass (designated as incremental value)
F	: Cutting feed rate
P	: Dwell time at hole bottom position

12 Programming Support Functions

12.1 Machining Method Support Functions

12.1.3.7 Fixed Cycle for Drilling (Type II)

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	-	-	-
L system	-	-	-	-	○	○	○	○

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

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

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

12 Programming Support Functions

12.1 Machining Method Support Functions

12.1.4 Mirror Image

12.1.4.1 Mirror Image by Parameter Setting

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
	M system	-	-	○	○	○	○	○	○
	L system	-	-	○	○	○	○	○	○

A parameter is used to designate the axis for which the mirror image function is to be executed before the machining program is run. When mirror image is set to ON by the parameter, an operation which is symmetrical on the left and right or at the top or bottom is performed. Each axis has its own parameter.

12.1.4.2 External Input Mirror Image

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
	M system	-	-	○	○	○	○	○	○
	L system	-	-	○	○	○	○	○	○

Signals from an external device (PLC) to request the mirror image operation either while a machining program is running or before it is run. When ON has been set for mirror image from an external device, an operation which is symmetrical on the left and right or at the top or bottom is performed. Each axis has its own request signal.

12 Programming Support Functions

12.1 Machining Method Support Functions

12.1.4.3 G Code Mirror Image

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	○	○	○	○
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.

G code	Function
G50.1	G code mirror image cancel
G51.1	G code mirror image ON

The program format for the G code mirror image is shown below.

```

G51.1 Xx1 Yy1 Zz1 ;
G51.1      : Mirror image on
Xx1, Yy1, Zz1 : Command axes and command positions
    
```

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

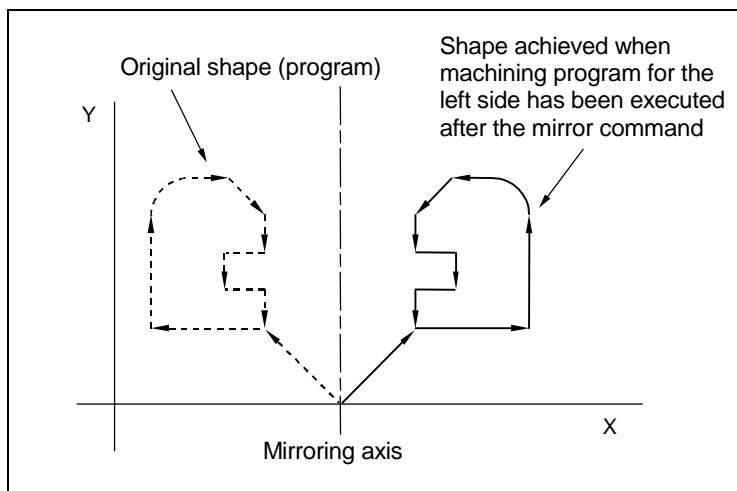
The program format for the G code mirror image cancel is shown below.

```

G50.1 Xx1 Yy1 Zz1 ;
G50.1      : Mirror image cancel
Xx1, Yy1, Zz1 : Command axes
    
```

The 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 Programming Support Functions

12.1 Machining Method Support Functions

12.1.4.4 Mirror Image for Facing Tool Posts

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	-	-	-
L system	-	Δ	Δ	Δ	-	Δ	Δ	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.

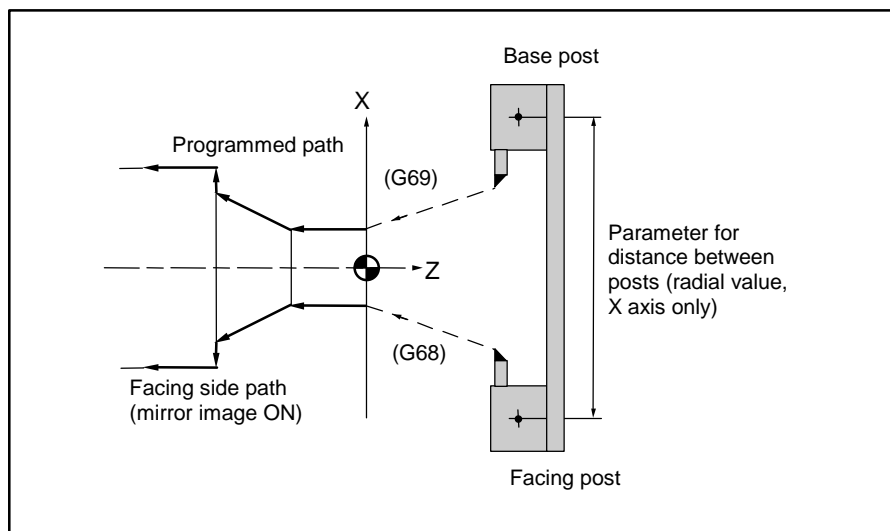
G code	Function
G68	Facing tool post mirror image ON
G69	Facing tool post mirror image OFF

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

When the G69 command is issued, the subsequent program coordinate systems are returned to the base side.

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

A parameter is used to set ON or OFF for the facing tool post mirror image function corresponding to the T commands.



12.1.4.5 T Code Mirror Image for Facing Tool Posts

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	-	-	-
L system	-	Δ	Δ	Δ	-	Δ	Δ	Δ

If, when tools that correspond to tool Nos. 1 to 64 are selected (T commands), these are tool Nos. for which the facing tool post mirror image function has already been designated with a parameter, the status equivalent to G68 (facing tool post mirror image ON) is established. When the commands apply to tool Nos. for which the facing tool post mirror image function is not designated, the status equivalent to G69 (facing tool post mirror image OFF) is established.

12 Programming Support Functions

12.1 Machining Method Support Functions

12.1.5 Coordinate System Operation

12.1.5.1 Coordinate Rotation by Program

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	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, 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.

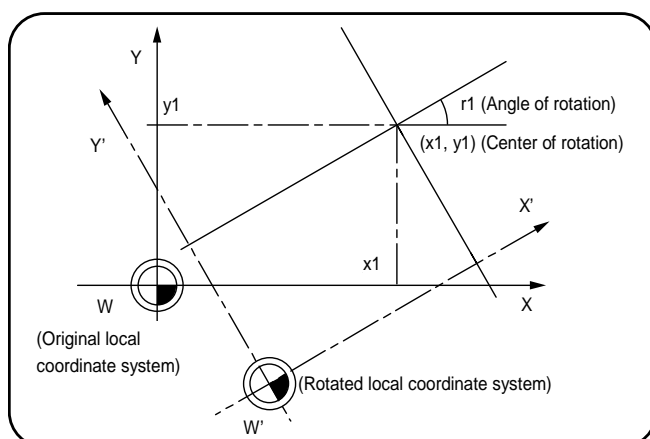
G68 Xx1 Yy1 Rr1 ; Coordinate rotation ON

G69 ; Coordinate rotation cancel

G68 : Call 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.

12 Programming Support Functions

12.1 Machining Method Support Functions

(Example)

```

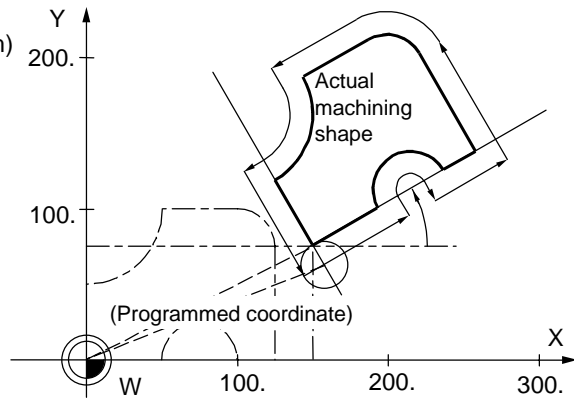
N01 G28 X Y Z ;
N02 G54 G52 X150. Y75. ; Local coordinate system assignment
N03 G90 G01 G42 X0 Y0 ; Tool radius compensation ON
N04 G68 X0 Y0 R30. ; Coordinate rotation ON
N05 M98 H101 ; Subprogram execution
N06 G69 ; Coordinate rotation cancel
N07 G54 G52 X0 Y0 ; Local coordinate system cancel
N08 G00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

12.1 Machining Method Support Functions

12.1.6 Dimension Input

12.1.6.1 Corner Chamfering / Corner R

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	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.

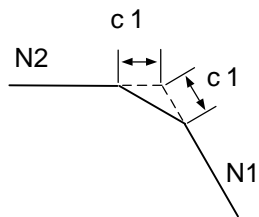
(1) Corner chamfering / Corner R I

When ",C" or ",R" is commanded for linear interpolation, corner chamfering or corner R can be inserted between linear blocks.

- **Corner chamfering**

Example:

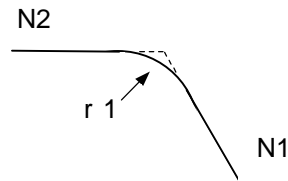
```
N1 G01 Xx1 Zz1, Cc1 ;
N2 Zz2 ;
```



- **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.

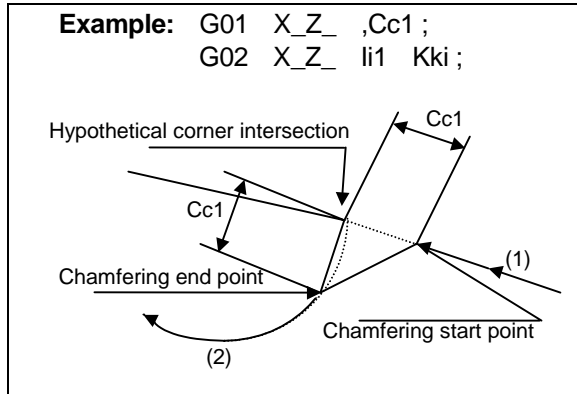
12 Programming Support Functions

12.1 Machining Method Support Functions

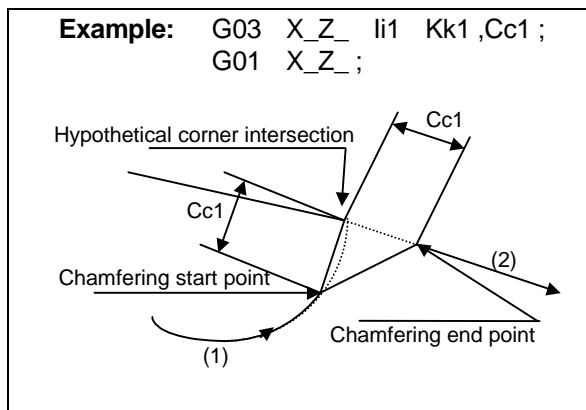
(2) Corner chamfering / corner R II (L system)

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

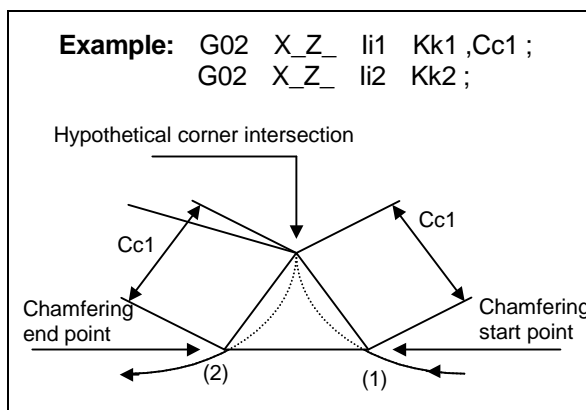
(a) Corner chamfering II (Linear – circular)



(b) Corner chamfering II (Circular - linear)



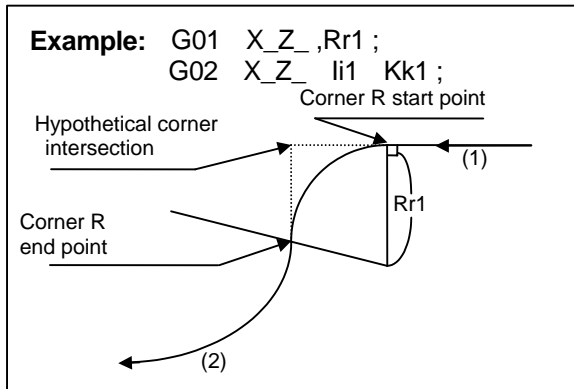
(c) Corner chamfering II (Circular - circular)



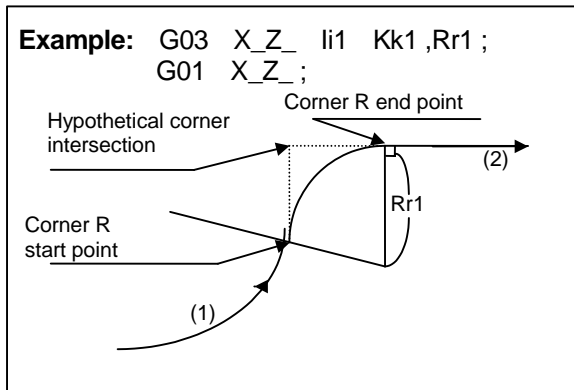
12 Programming Support Functions

12.1 Machining Method Support Functions

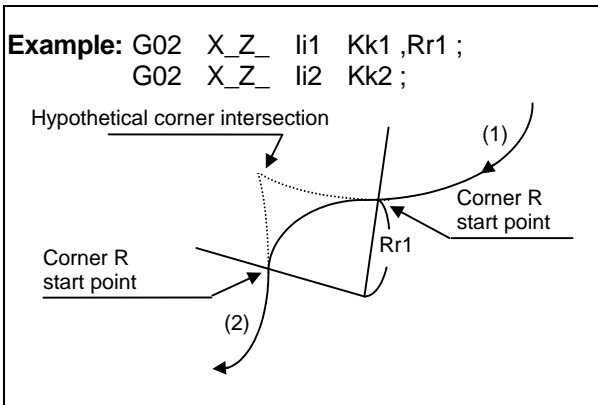
(d) Corner R II (Linear - circular)



(e) Corner R II (Circular – linear)



(f) Corner R II (Circular – circular)



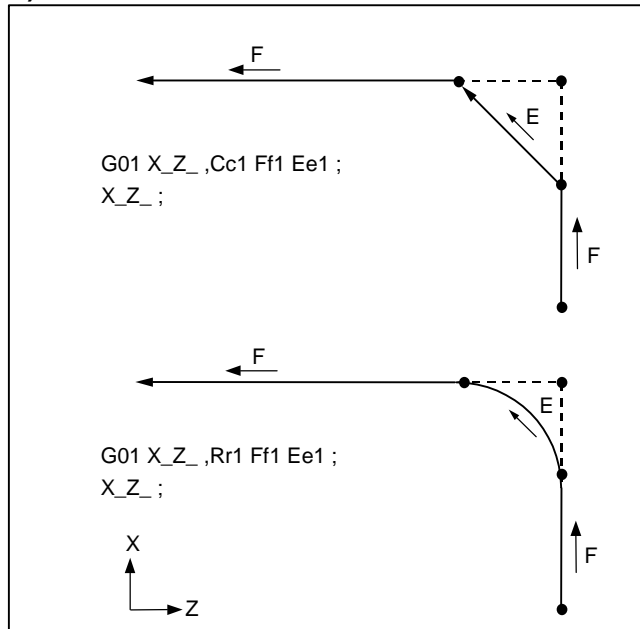
12 Programming Support Functions

12.1 Machining Method Support Functions

(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 Programming Support Functions

12.1 Machining Method Support Functions

(4) Corner chamfering / corner R (I, K designation)

With this command format, by means of parameter settings, corners are chamfered using the "I", "K" or "C" address without a comma, and corners are rounded using the "R" address.

The ",C" and ",R" commands with commas can also be used.

This command format is valid for the L system of M64AS, M64S, M65S, and M66S only.

(a) Corner chamfering (I, K designation)

Corners are chamfered using the "I_", "K_" or "C_" address with no comma. Corners can be chamfered to any angle. Signs, if they are provided for the corner chamfering commands, are ignored.

Command format

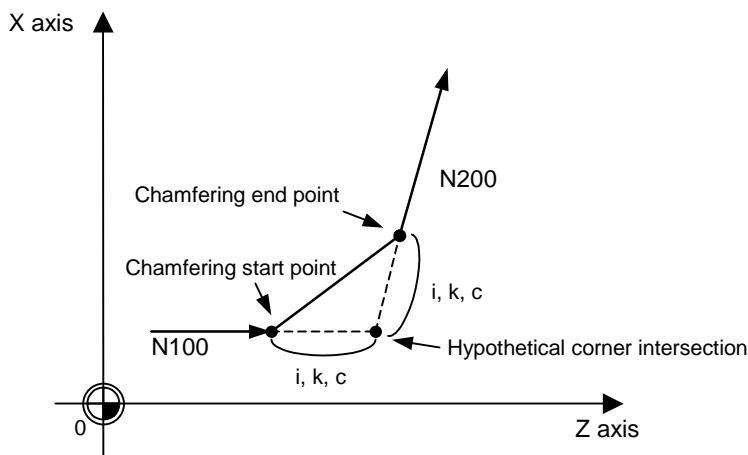
N100 Xx/Uu Zz/Ww Ii/Kk/Cc ;

N200 Xx/Uu Zz/Ww ;

X/u : X-axis end point coordinate

Z/w : Z-axis end point coordinate

i/k/c : The length from the hypothetical corner intersection to the chamfering start point or chamfering end point is designated using the I, K or C address.



- If multiple "I", "K" or "C" addresses or duplicated addresses have been designated in the same block, the last command will take effect.
- If both corner chamfering and corner R commands are present in the same block, the last command will take effect.
- If "C" is used as the name of an axis, corner chamfering commands cannot be designated using the "C" address.
- If "C" is used as a 2nd miscellaneous function, corner chamfering commands cannot be designated using the "C" address.
- Corner chamfering commands using the "I" or "K" address cannot be designated in an arc command block. "I" and "K" are the arc center commands.

12 Programming Support Functions

12.1 Machining Method Support Functions

(b) Corner R (I, K designation)

Corners are rounded using the "R_" address with no comma. Corners can be rounded to any angle. Signs, if they are provided for the corner R commands, are ignored.

Command format

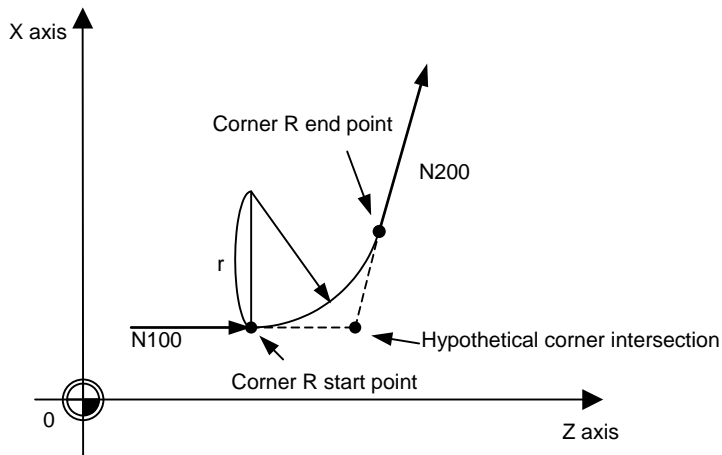
```
N100 Xx/Uu Zz/Ww Rr ;
```

```
N200 Xx/Uu Zz/Ww ;
```

x/u : X-axis end point coordinate

z/w : Z-axis end point coordinate

r : Radius of corner R arc



- If both corner chamfering and corner R commands are present in the same block, the last command will take effect.
- Corner R commands using the "R" address cannot be designated in an arc command block. "R" is the arc radius command.

12 Programming Support Functions

12.1 Machining Method Support Functions

12.1.6.2 Linear Angle Command

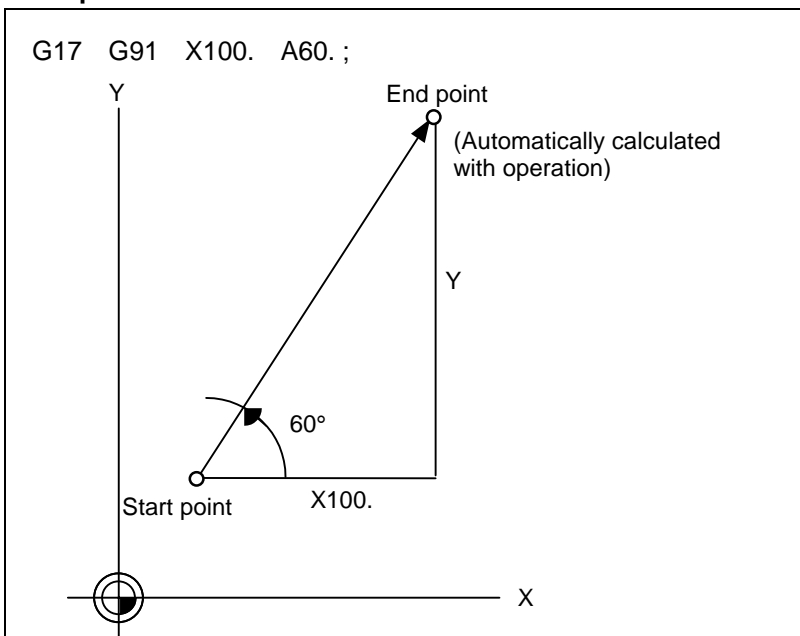
M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ

The end point coordinates are automatically calculated by assigning one element (one component of the selected plane) of the end point coordinates and the linear angle.

G17 Xx1 Aa1 ; or G17 Yy1 Aa1 ;

G17 : Plane selection
 Xx1, Yy1 : 1 element
 Aa1 : Angle

Example



Note 1: If both the axis "A" or 2nd miscellaneous function "A" are used, address "A" is treated as the axis "A" command and or the 2nd miscellaneous function, respectively.

12 Programming Support Functions

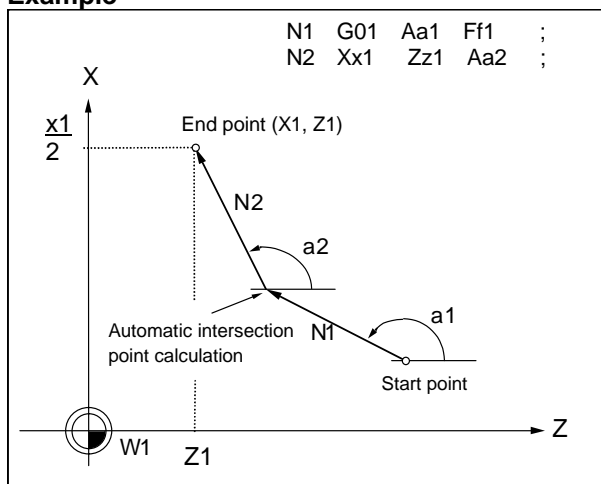
12.1 Machining Method Support Functions

12.1.6.3 Geometric Command

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	Δ	Δ	Δ	Δ
L system	○	○	○	○	○	○	○	○

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.

12 Programming Support Functions

12.1 Machining Method Support Functions

(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 : Incremental coordinates from circular end point
P and Q : Absolute center coordinates of circular arc
H = 0 : Intersection with shorter line
H = 1 : Intersection with longer line

12 Programming Support Functions

12.1 Machining Method Support Functions

(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 : Incremental coordinates from circular end point
 P and Q : Absolute center coordinates of circular arc (L3 only)
 H = 0 : Intersection with shorter 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 ;
    
```

12 Programming Support Functions

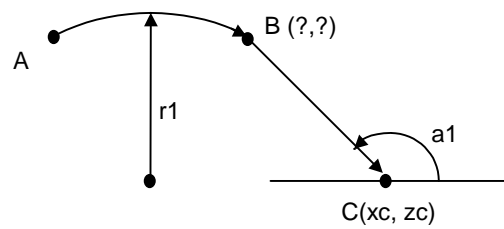
12.1 Machining Method Support Functions

(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 Programming Support Functions

12.1 Machining Method Support Functions

12.1.6.4 Polar Coordinate Command

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	Δ	Δ	Δ	Δ
L system	-	-	-	-	-	-	-	-

With this function, the end point coordinate value is commanded with the radius and angle.

Command format

G16 ; Polar coordinate command mode ON

G15 ; Polar coordinate command mode OFF

Example of program

<p>G1x ; G16 ; G9x G01 Xx1 Yy1 F2000 ; :</p>	<p>Plane selection for polar coordinate command (G17/G18/G19) Polar coordinate command mode ON Polar coordinate command G9x : Center selection for polar coordinate command (G90/G91) G90···The workpiece coordinate system zero point is the polar coordinate center. G91···The present position is the polar coordinate center. x1 : 1st axis for the plane···The radius commanded y1 : 2nd axis for the plane···The angle commanded</p>
---	---

For G90/G17(X-Y plane)

<p>G15 ;</p>	<p>Polar coordinate command mode OFF</p>
--------------	--

12 Programming Support Functions

12.1 Machining Method Support Functions

12.1.7 Axis Control

12.1.7.1 High-speed Machining

12.1.7.1.3 High-speed Machining Mode III

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	Δ 16m	Δ 16m	Δ 16m	Δ 16m	Δ 16m	Δ 16m	Δ 16m
L system	-	-	-	-	-	-	-	-

This function runs machining programs, in which free-form curved surfaces have been approximated by fine-segments, at high speed.

It is effective in increasing the speed at which dies with free-form curved surfaces are machined. High-accuracy machining can be achieved by using this function in combination with the high-accuracy control function.

Command format

G05 P3 ; High-speed machining mode III ON G05 P0 ; High-speed machining mode III OFF

Example of program

<pre> G28 X0. Y0. Z0. ; G91 G00 X-100. Y-100. ; G01 F3000 ; G05 P3 ; X0.1 ; X0.1 Y0.001 ; X0.1 Y0.002 ; G90 G00 X0 Y0 Z0 ; G02 X__Y__R__F__ ; G03 X__Y__I__J__F__ ; . . . G05 P0 ; M02; </pre>	<div style="display: flex; align-items: center;"> <div style="margin-right: 10px;"> <p>.....</p> <p>} High-speed machining mode III ON</p> <p>} Incremental modal linear cutting</p> <p>} Absolute modal Rapid traverse R-designated arc Arc</p> <p>..... High-speed machining mode III OFF</p> </div> </div>
--	---

- (1) The following G codes can be used in the high-speed machining mode III:
G00, G01, G02, G03, G90, G91, G17, G18 and G19.
- (2) In the high-speed machining mode III, only the axis names, movement amounts (no variables or arithmetic operations), F commands and I/J/K/R/P commands can be designated. (Comments can be used.)
- (3) The machining speed may be compromised depending on the number of characters in a block.

12 Programming Support Functions

12.1 Machining Method Support Functions

12.1.7.1.4 High-speed High-accuracy Control 1

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	Δ 16m	Δ 16m	-	Δ 16m	Δ 16m	Δ 16m
L system	-	-	-	-	-	-	-	-

This function runs machining programs, in which free-form curved surfaces have been approximated by fine-segments, at high speed and with a high accuracy.

It is effective in increasing the speed at which dies with free-form curved surfaces are machined.

Command format

G05.1 Q1	;	High-speed high-accuracy control I ON
G05.1 Q0	;	High-speed high-accuracy control I OFF

The performance of simultaneous 3 axis and 1mm-length fine-segment is as follows.

High-accuracy control I mode	Performance of fine-segment length		Program restriction
	No radius compensation	Radius compensation	
Invalid	8.4m/min	8.4m/min	None
Valid	16.8m/min	16.8m/min	Available

Program restrictions

The machining speed may drop depending on the number of characters in one block.

During high-speed high-accuracy control I operation using tape operation, the machining speed may be suppressed depending on the program transmission speed and the number of characters in one block.

Modes which can be designated by commands

The modal statuses when the G05.1 Q1 command is assigned must be as shown in the table below. If the conditions listed are not satisfied, a program error will result.

Function	G code modal
Tool radius compensation mode	G40
Tool length compensation	G49
G code mirror image	G50.1
Mirror image by parameter setting	Cancel
External signal mirror image	Cancel
Cutting mode	G64
Macro modal call mode	G67
Coordinate conversion mode	G69
Fixed cycle mode	G80
Per-revolution feed	G94
Constant surface speed control mode	G97
Interrupt-type macro mode	M97

12 Programming Support Functions

12.1 Machining Method Support Functions

12.1.7.1.5 High-speed High-accuracy Control 2

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	Δ 33m	Δ 67m(*)	-	Δ 33m	Δ 67m(*)	Δ 67m(*)
L system	-	-	-	-	-	-	-	-

(Note) (*) changes depending on the axis configuration and operation condition.

This function runs machining programs, in which free-form curved surfaces have been approximated by fine-segments, at high speed and with a high accuracy. It is effective in increasing the speed at which dies with free-form curved surfaces are machined.

Command format

G05 P10000 ;	High-speed high-accuracy control II ON
G05 P0 ;	High-speed high-accuracy control II OFF

The performance of fine-segment execution when 3 axes are moved simultaneously is as follows.

High-accuracy control II mode	Performance of fine segment execution (1mm-length) (No radius compensation)		Program restriction
	Fairing invalid	Fairing valid	
Invalid	8.4m/min	8.4m/min	None
Valid (Number of NC axes 1 to 4)	67.5m/min	50.6m/min	Available
Valid (Number of NC axes 5 and 6)	50.6m/min	42.1m/min	

Program restrictions

The machining speed may drop depending on the number of characters in one block.

During high-speed high-accuracy control II operation using tape operation, the machining speed may be suppressed depending on the program transmission speed and the number of characters in one block.

When the fairing is valid (#8033 is set to 1), if the successive fairing is executed in the machining program, the performance of fine-segment execution may decelerate less than the value described in the above table. In the network connection, the value described in the above table may not be guaranteed depending on the state.

Modes which can be designated by commands

The modal statuses when the G05 P10000 command is assigned must be as shown in the table below. If the conditions listed are not satisfied, a program error will result.

Function	G code modal
Tool radius compensation mode	G40
Tool length compensation	G49
G code mirror image	G50.1
Mirror image by parameter setting	Cancel
External signal mirror image	Cancel
Cutting mode	G64
Macro modal call mode	G67
Coordinate conversion mode	G69
Fixed cycle mode	G80
Per-revolution feed	G94
Constant surface speed control mode	G97
Interrupt-type macro mode	M97

12 Programming Support Functions

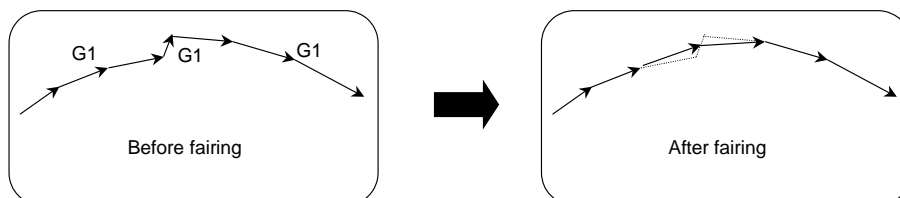
12.1 Machining Method Support Functions

Additional functions when high-speed high-accuracy control 2 mode is ON

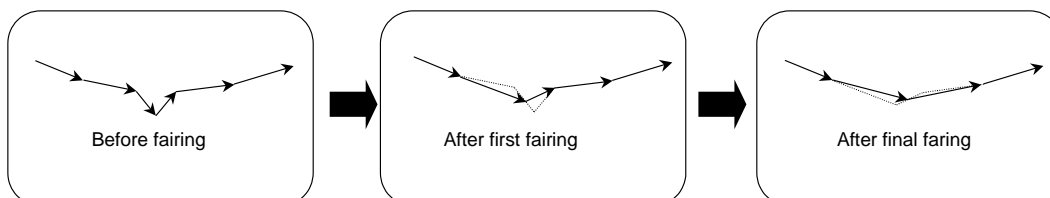
(1) Fairing

If the protruding path (zigzagging path) is shorter than the parameter setting values in the machining program generated with a CAM, etc., this function can be used to eliminate the protruding paths smaller than the setting value so that the front and back paths are smoothly connected.

This function is valid only for continuous linear commands (G1).

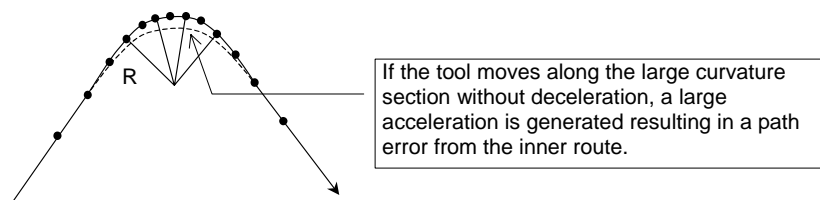


If there is any protruding path after fairing, fairing is repeated.



(2) Acceleration clamp speed

Parameter settings are used to clamp the speed so that the acceleration occurring due to movement of each block does not exceed the tolerable value. This function clamps the speed optimally even at a section where "angle change at each block is small but entire curvature is large" such as shown below. The tolerable acceleration value is calculated from the parameter setting values.



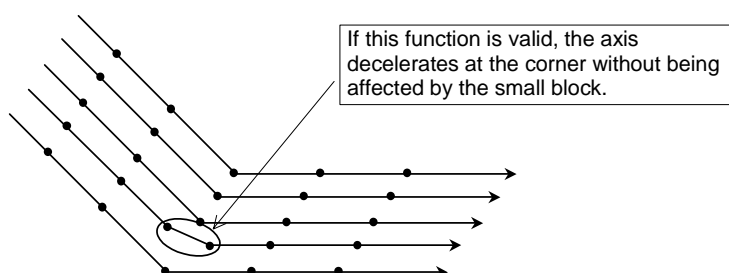
(3) High-speed mode corner deceleration

During high-accuracy control, if the angle is large between adjacent blocks in the machining program, this function automatically decelerates so that the acceleration generated when passing through the corner is within the tolerable value.

If a small block is inserted at the corner section with the machining program generated with the CAM, etc., the corner passing speed will not match the periphery. This can affect the machining surface.

If this type of small block is inserted while this function is valid, the corner will be largely judged by the parameter settings.

The small block is excluded when the angle is judged, but it is not excluded from the actual movement command.



12 Programming Support Functions

12.1 Machining Method Support Functions

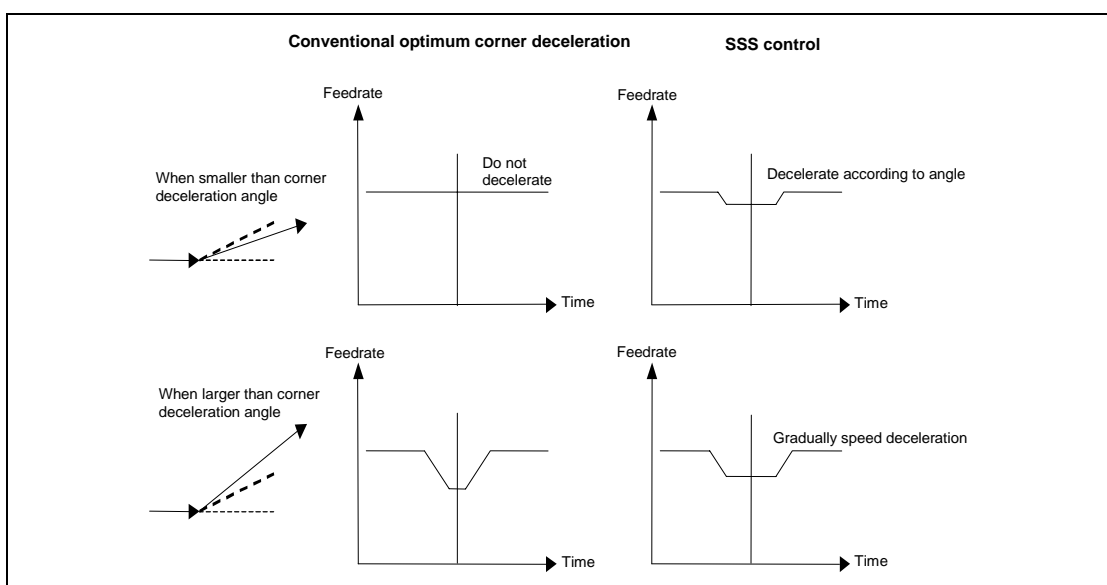
12.1.7.1.7 SSS Control

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	-	Δ	Δ
L system	-	-	-	-	-	-	-	-

(Note) The "12.1.7.1.5 High-speed high-accuracy control 2" option is required to use this function.

With normal high-accuracy control, the angle between two blocks is compared with the corner deceleration angle to determine whether to execute corner deceleration between the blocks. This can cause the speed to suddenly change between the blocks with an angle close to the corner deceleration angle, or can cause scratches or streaks to form.

With SSS (Super Smooth Surface) control, the large area path information is used instead of just the angle between the blocks. Thus, optimum speed control that is not adversely affected by minute steps or waviness is possible. This enables machining with fewer scratches and streaks on the cutting surface compared to the normal high-accuracy control function.



[Features of SSS control]

- (1) Speed fluctuation, caused by the effect of the blocks (minute stepping or waviness) and resulting in disturbed machining, is suppressed, and scratches caused by these blocks is reduced.
- (2) This speed control is not easily affected by the steps found in the path.
- (3) If adjacent paths are similar in terms of shape accuracy, the acceleration/deceleration pattern is also similar, and the fluctuation of accuracy caused by a difference in speed is suppressed.
- (4) Even if corner deceleration is not required, the speed is clamped if the predicted acceleration is high.
- (5) The machining time can be reduced in machining applications having many corners.

The length of the path direction recognized with SSS control can be adjusted with the machining parameter "reference length". The range is increased as the setting value increases, and the effect of the error is reduced.

(Note) The high-speed high-accuracy control 2 option is required to use this function.

12 Programming Support Functions

12.1 Machining Method Support Functions

12.1.7.2 Chopping

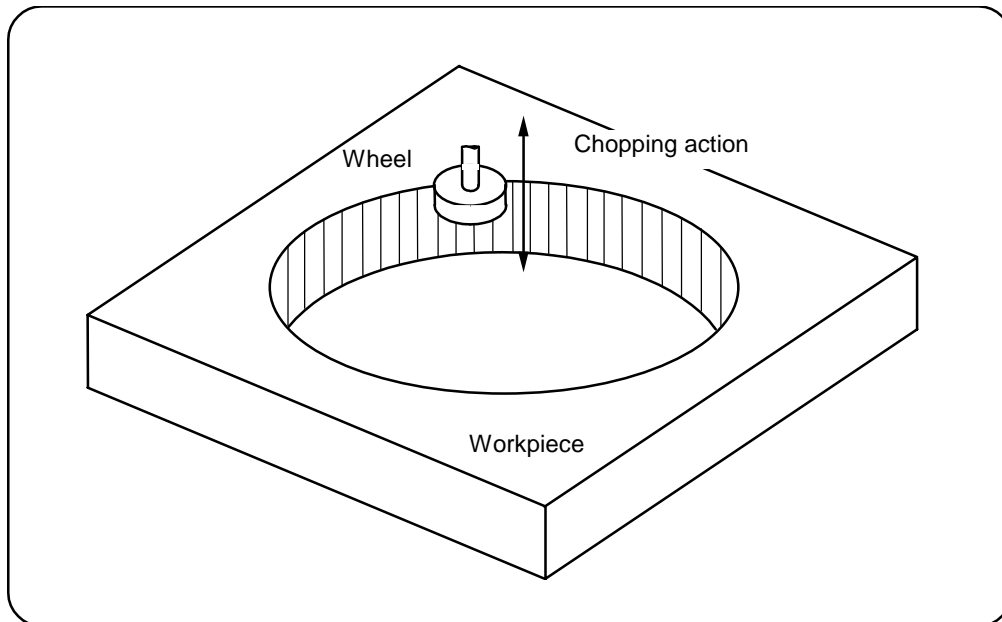
12.1.7.2.1 Chopping

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	Δ	Δ	Δ
L system	-	-	-	-	-	Δ	Δ	Δ

This function continuously raises and lowers the chopping axis independently of the program operation when workpiece profiles are to be cut. It can be used for grinding operations using machining centers, for instance.

Which of the axes is to serve as the chopping axis is set by a parameter ahead of time.

(1) Chopping action



12 Programming Support Functions

12.1 Machining Method Support Functions

The chopping operation is initiated by setting the upper dead point position, lower dead point position and number of cycles (number of up/down movements per minute) and pressing the chopping start switch.

Note 1: The upper dead point position, lower dead point position and number of cycles are set and the start and stop commands are designated by input signals from the user PLC.

Note 2: The setting for the number of cycles differs according to the motor, inertia and other factors.

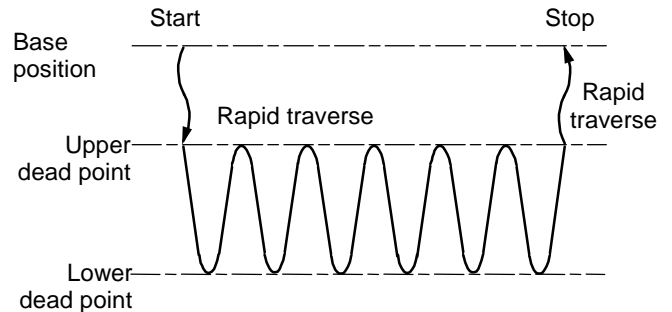
The chopping operation is performed as follows.

- The axis moves from the base position to the upper dead point by rapid traverse.
- Next, the axis moves repeatedly from the upper dead point to the lower dead point and then from the lower dead point to the upper dead point. (Sinusoidal waveforms)
The feed rate is tailored to achieve the number of cycles set for the up/down motion.

Chopping override

Override in 1% increments from 0% to 100% can be applied to the chopping operation.

Note: Bear in mind that the override increment differs according to the machine specifications.



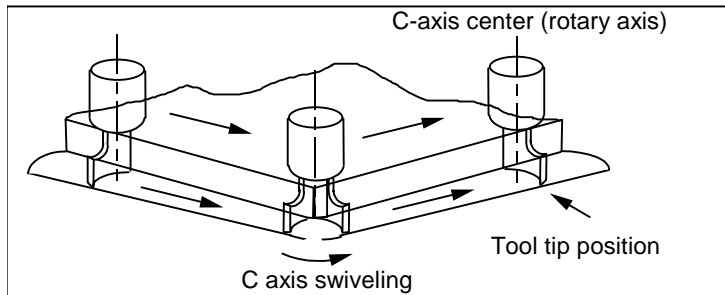
12 Programming Support Functions

12.1 Machining Method Support Functions

12.1.7.4 Normal Line Control

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	Δ	Δ	Δ
L system	-	-	-	-	-	-	-	-

This function controls the swiveling of the C axis (rotary axis) so that the tool is always pointing in the normal line direction for the X and Y axes movement commands during program operation. It can be used for machining springs, for sewing operations involving sewing machines, carpets, etc. It is valid for only one C axis (rotary axis).



G code	Function
G40.1	Normal line direction control cancel
G41.1	Normal line direction control left ON
G42.1	Normal line direction control right ON

12 Programming Support Functions

12.1 Machining Method Support Functions

12.1.7.5 Circular Cutting

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	-	-	-	-	-	-	-	-

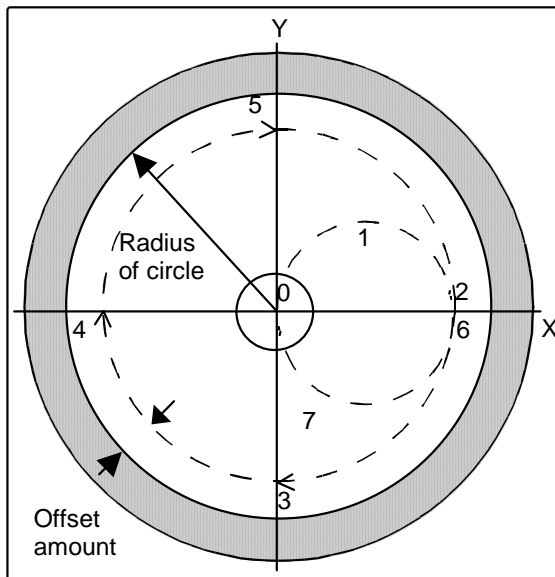
In circular cutting, a system of cutting steps are performed: first, the tool departs from the center of the circle, and by cutting along the inside circumference of the circle, it draws a complete circle, then it returns to the center of the circle. The position at which G12 or G13 has been programmed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

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

The program format is given below.

G12/13 li Dd Ff ;

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



When the G12 command is used
 (path of tool center)

0 → 1 → 2 → 3 → 4 → 5 → 6 → 7 → 0

When the G13 command is used
 (path of tool center)

0 → 7 → 6 → 5 → 4 → 3 → 2 → 1 → 0

(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 Programming Support Functions

12.1 Machining Method Support Functions

12.1.8 Multi-part System Control

12.1.8.1 Synchronization between Part Systems

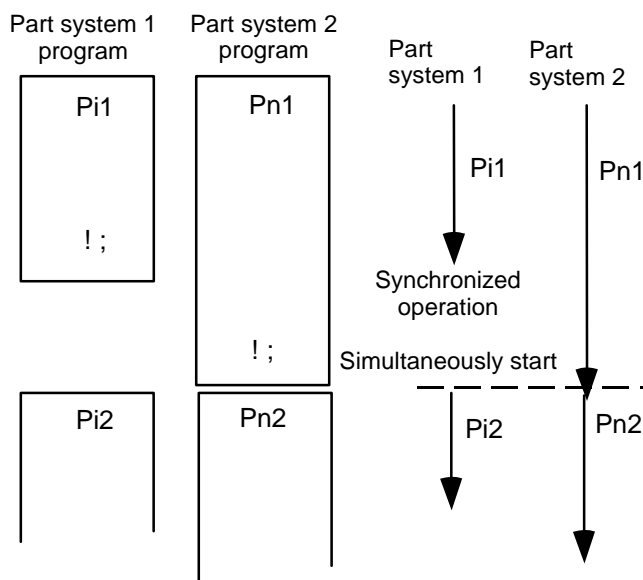
M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	Δ	Δ	Δ	-	-	-	-
L system	-	Δ	Δ	Δ	-	○	○	○

This function enables part systems to be synchronized (block start synchronization) using the program.

! ; Part system 1 and part system 2 are synchronized.

When the part system number is to be omitted

This synchronization command makes it possible to control the simultaneous operation of two part systems and the independent operation of part systems.



(Note 1) When the part system with which the operation is to be synchronized is not operating, the other part system moves on to the next block. This function is useful when conducting program checks on a part system by part system basis.

(Note 2) The synchronization command is normally assigned as a separate command. However, if a movement command and M, S or T command have been assigned in the same block, a parameter is set to decide whether the operations are to be synchronized after the movement command and M, S or T command have been executed or the movement command and M, S or T command are to be executed upon completion of the synchronization.

12 Programming Support Functions

12.1 Machining Method Support Functions

12.1.8.2 Start Point Designation Synchronization

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	△	△	△	-	-	-	-
L system	-	△	△	△	-	○	○	○

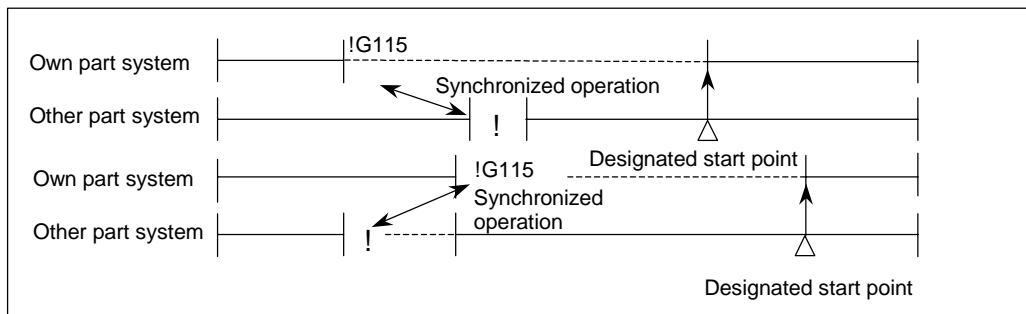
The synchronizing point can be placed in the middle of the block by designating the start point.

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

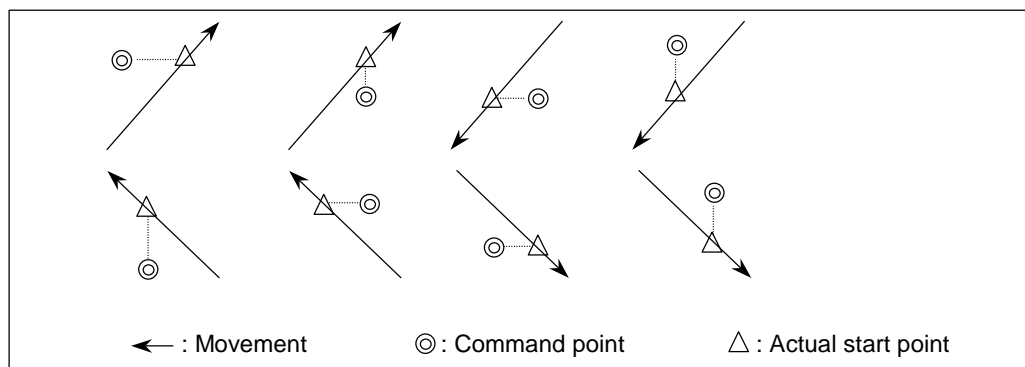
Command format

!LI G115 X_ Z_ C_ ;	
!LI	: Synchronizing command
G115	: G command
X_,Z_,C_	: 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.



12 Programming Support Functions

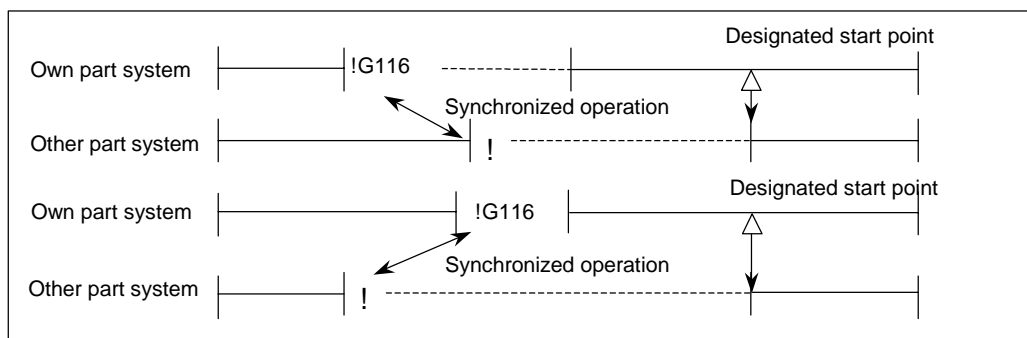
12.1 Machining Method Support Functions

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

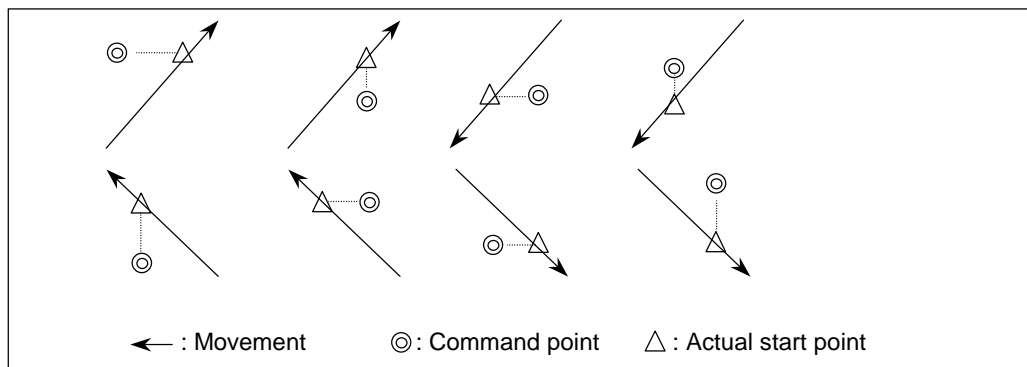
Command format

!LI G116 X_ Z_ C_ ;	
!LI	: Synchronizing command
G116	: G command
X_,Z_,C_	: 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 other part system has reached all of the start point axis coordinates.



12 Programming Support Functions

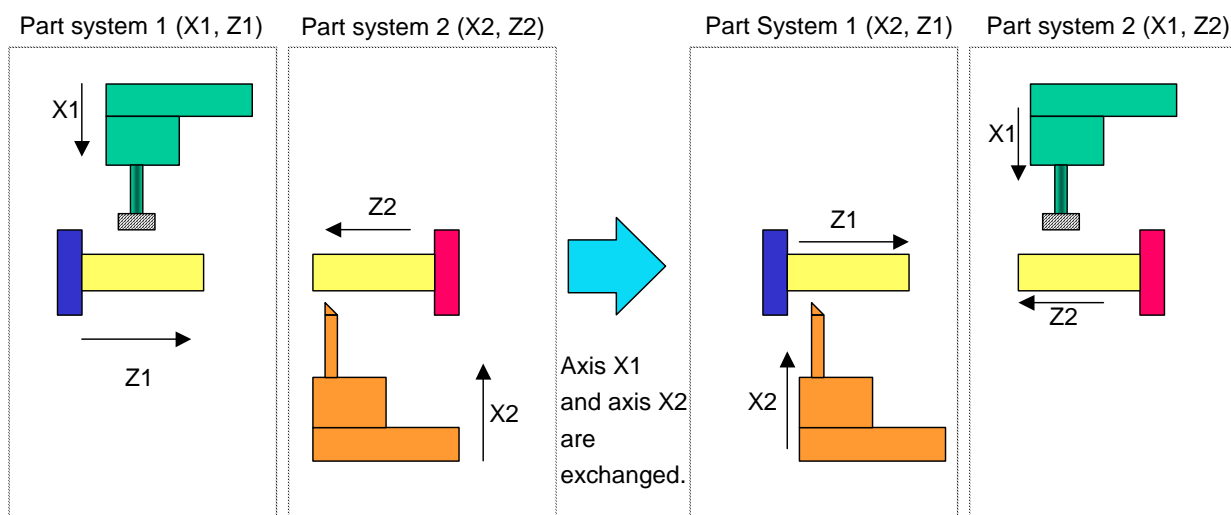
12.1 Machining Method Support Functions

12.1.8.3 Cross Machining Command

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	-	-	-
L system	-	-	-	-	-	-	Δ	Δ

This function enables any axis to be replaced by another axis between part systems. This makes it possible to perform operations which are not possible with regular axis configurations: for instance, tools which are provided only on part system 1 can be used for machining on part system 2.

This manual contains descriptions where part system numbers have been added after the axis names in order to identify which part system the axes belong to (such as X1). In terms of designating the program commands, however, a single letter is used to designate the axis address just as before.



Cross machining control is switched from ON to OFF and vice versa using a PLC signal. Parameters are used to set what axis is to be placed in the cross machining control status in response to which PLC signal.

12 Programming Support Functions

12.1 Machining Method Support Functions

12.1.8.5 Control Axis Synchronization

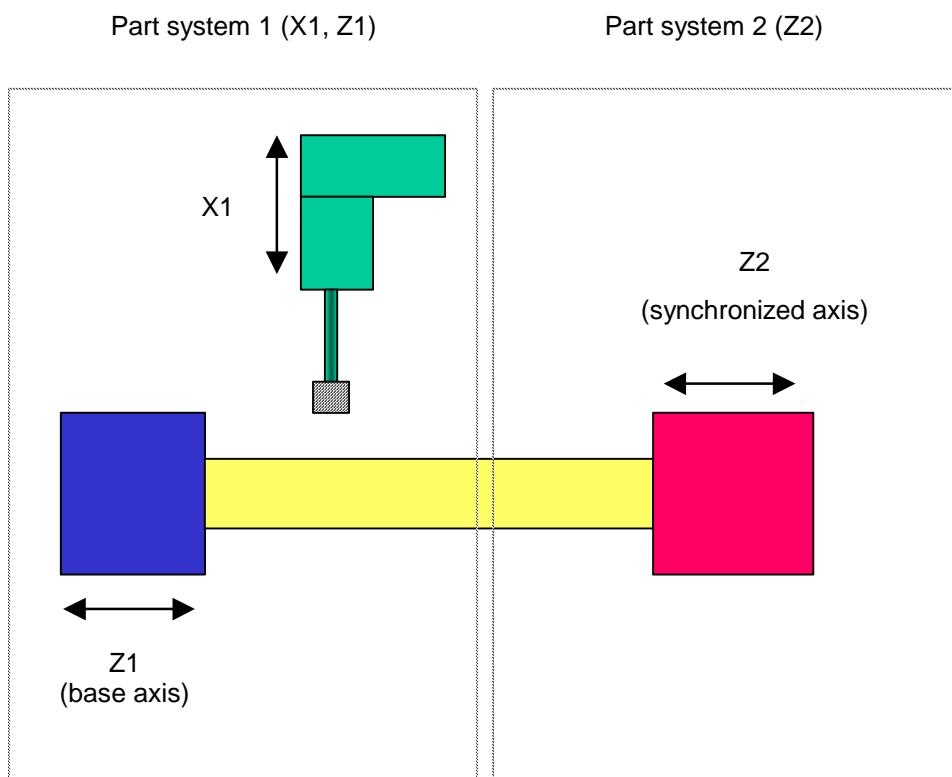
M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	-	-	-
L system	-	-	-	-	-	-	Δ	Δ

Synchronization control enables an arbitrary control axis in the other part system to move in synchronization with the movement command assigned to an arbitrary control axis.

The direction in which the axis is to move synchronously can also be reversed using a parameter.

Base axis :Axis to which movement command to synchronize axes is assigned.

Synchronized axis :Axis whose movement is synchronized with base axis.



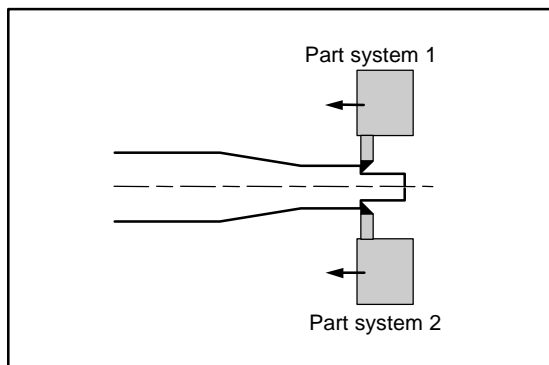
12 Programming Support Functions

12.1 Machining Method Support Functions

12.1.8.6 Balance Cut

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	-	-	-
L system	-	Δ	Δ	Δ	-	Δ	Δ	Δ

When workpiece that is relatively long and thin is machined on a lathe, deflection may result, making it impossible for the workpiece to be machined with any accuracy. In cases like this, the deflection can be minimized by holding tools simultaneously from both sides of the workpiece and using them in synchronization to machine the workpiece (balance cutting). This method has an additional advantage: since the workpiece is machined by two tools, the machining time is reduced. 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 Part system 2 program

<pre>T0101; G00 X_ Z_; G15; G01 Z_ F0.4; ⋮</pre>
--

<pre>T0102; G00 X_ Z_; G15; G01 Z_ F0.4; ⋮</pre>
--

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

12 Programming Support Functions

12.1 Machining Method Support Functions

12.1.8.7 Common Memory between Tool Posts

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	-	-	-
L system	-	-	-	-	-	○	○	○

For the machine with two part systems, the common variables and tool compensation memory which exist in 1-part system and 2-part system respectively can be made common in those systems by the parameter setting.

(1) Common variables

For 2-part system, normally, common variables #100 to #199 are provided per part system, and common variables #500 to #999 are common for part systems. However, when this function is valid, common variables #500 to #999 are also provided per part system.

To make a part or all of common variables #100 to #199 and #500 to #999 to be used commonly between part systems, set parameters to designate the number of variables to be used commonly for #100 to #199 and #500 to #999 respectively.

As for the variables designated to be common to part systems, the data for part system 1 is used.

(2) Tool compensation memory

When the common tool compensation memory between part systems is valid, the tool compensation memory used during operation in part system 2 is changed so that part system 1 data is used. Therefore, part system 2 data is not used because part systems are made common.

This function is only for the lathe system and supported up to 2 part systems.

(Note) The setting (such as number of axes and axis names) for the tool offset amount of part system 1 and part system 2 must be the same.

If the settings differ, the setting of part system 1 will be applied.

12.1.8.9 2-part System Program Management

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	-	-	-
L system	-	-	-	-	-	○	○	○

Using parameter (#1050 MemPrg) the machining program can be created for each system, and can be input/output for each system.

12 Programming Support Functions

12.1 Machining Method Support Functions

12.1.9 Data Input by Program

12.1.9.1 Parameter Input by Program

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	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 L50 ; Data setting command									
P	<u>Major classification No.</u>	A	<u>Axis No.</u>	N	<u>Data No.</u>	H□	<u>Bit type data</u> ;	}	Parameter settings in data setting mode
P	<u>Major classification No.</u>	A	<u>Axis No.</u>	N	<u>Data No.</u>	D	<u>Byte type data</u> ;		
P	<u>Major classification No.</u>	A	<u>Axis No.</u>	N	<u>Data No.</u>	S	<u>Word type data</u> ;		
P	<u>Major classification No.</u>	A	<u>Axis No.</u>	N	<u>Data No.</u>	L	<u>2-word type data</u> ;		
G11 ; Data setting mode cancel (data setting completed)									

The following types of data formats can be used according to the type of parameter (axis-common and axis-independent) and data type.

With axis-common data

Axis-common bit-type parameter ----- P__ N__ H□__ ;
 Axis-common byte-type parameter ----- P__ N__ D__ ;
 Axis-common word-type parameter ----- P__ N__ S__ ;
 Axis-common 2-word-type parameter ----- P__ N__ L__ ;

With axis-independent data

Axis-independent bit-type parameter ----- P__ A__ N__ H□__ ;
 Axis-independent byte-type parameter ----- P__ A__ N__ D__ ;
 Axis-independent word-type parameter ----- P__ A__ N__ S__ ;
 Axis-independent 2-word-type parameter ----- P__ A__ N__ L__ ;

(Note 1) The order of addresses in a block must be as shown above.

(Note 2) For a bit type parameter, the data type will be H□ (□ is a value between 0 and 7).

(Note 3) The axis number is set in the following manner: 1st axis is "1", 2nd axis is "2", and so forth.

When using the 2-part system, the 1st axis in each part system is set as "1", the 2nd axis is set as "2", and so forth.

(Note 4) Command G10L50 and G11 in independent blocks. A program error will occur if not commanded in independent blocks.

Depending on the G90/G91 modal status when the G10 command is assigned, the data is used to overwrite the existing data or added.

12 Programming Support Functions

12.1 Machining Method Support Functions

12.1.9.2 Compensation Data Input by Program

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ

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

12 Programming Support Functions

12.1 Machining Method Support Functions

(2) Tool offset input

The tool offset 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 offset amount
G10 L11	Tool length wear offset amount
G10 L12	Tool radius shape offset amount
G10 L13	Tool radius wear offset amount

The tool offset input format is as follows.

G10 LI1 Pp1 Rr1 ;	
G10 LI1	: Command for setting offset amount
Pp1	: Offset No.
Rr1	: Offset amount

(Note) When LI1 has been omitted, the tool length shape offset amount is set. Omitting Pp1 results in a program error.

[L system]

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

The tool offset input format is as follows.

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

12 Programming Support Functions

12.1 Machining Method Support Functions

12.1.10 Machining Modal

12.1.10.1 Tapping Mode

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		○	○	○	○	○	○	○	○
	M system	○	○	○	○	○	○	○	○
	L system	○	○	○	○	○	○	○	○

When tapping mode commands are issued, the NC system is set to the following internal control modes required for tapping.

1. Cutting override is fixed at 100%.
2. Deceleration commands at joints between blocks are invalid.
3. Feed hold is invalid.
4. Single block is invalid.
5. "In tapping mode" signal is output.

G code	Function
G63	Tapping mode ON

The tapping mode command will be canceled with the following commands:

- Exact stop check mode (G61)
- Automatic corner override (G62)
- Cutting mode (G64)
- 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

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		○	○	○	○	○	○	○	○
	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 Programming Support Functions

12.2 Machining Accuracy Support Functions

12.2 Machining Accuracy Support Functions

12.2.1 Automatic Corner Override

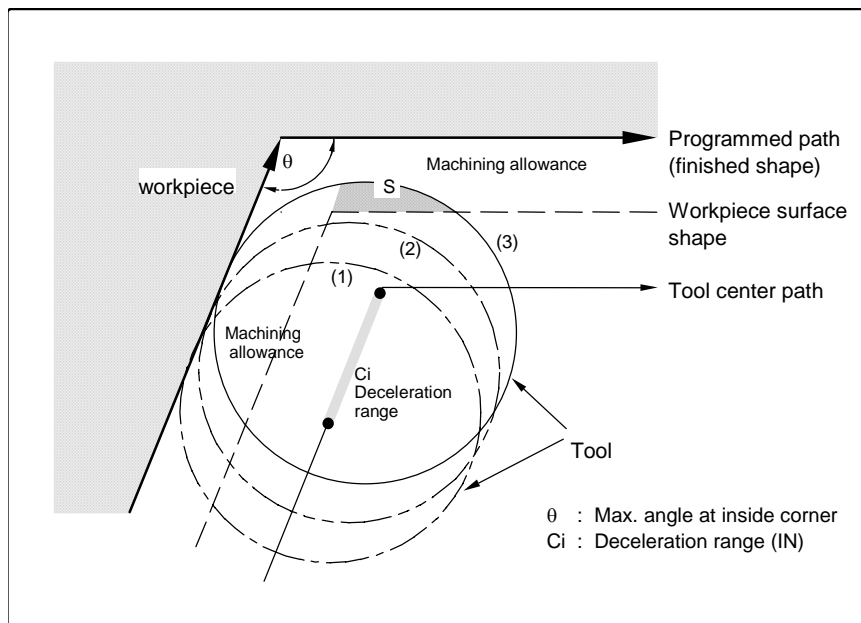
M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

To prevent machining surface distortion due to the increase in the cutting load during cutting of corners, this function automatically applies an override on the cutting feed rate so that the cutting amount is not increased for a set time at the corner.

Automatic corner override is valid only during tool radius compensation.

The automatic corner override mode is set to ON by the G62 command and it is canceled by any of the G commands below.

- G40 Tool radius compensation cancel
- G61 Exact stop check mode
- G63 Tapping mode
- G64 Cutting mode
- 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 C_i .

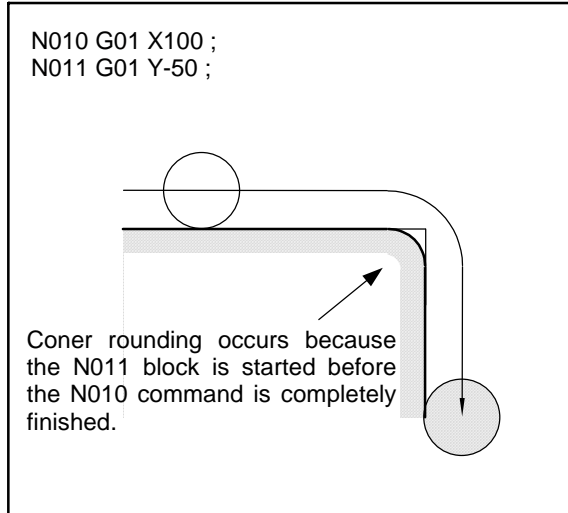
12 Programming Support Functions

12.2 Machining Accuracy Support Functions

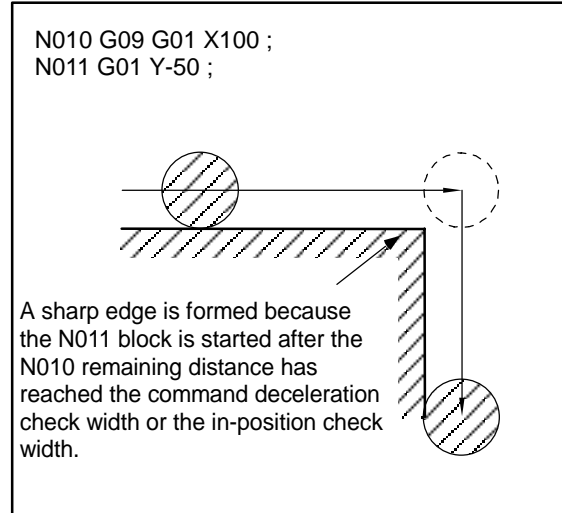
12.2.2 Deceleration Check

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 feed rate of the control axis changes suddenly.

Without deceleration check



With deceleration check



The conditions for executing deceleration check are described below.

(1) Deceleration check in the rapid traverse mode

In the rapid traverse mode, the deceleration check is always performed when block movement is completed before executing the next block.

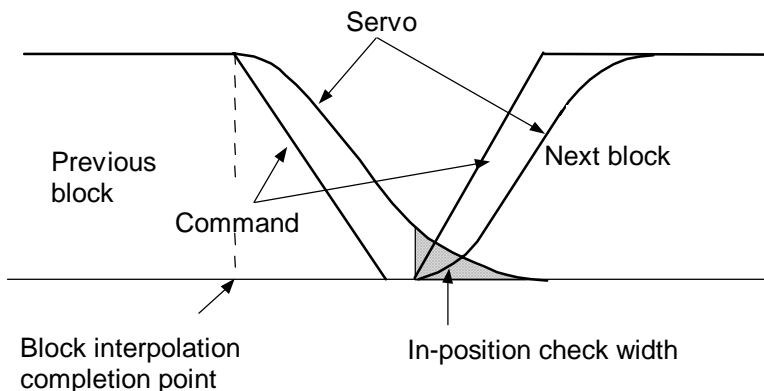
(2) Deceleration check in the cutting feed mode

In the cutting feed mode, the deceleration check is performed at the end of block when any of the conditions below is applicable before executing the next block.

- (a) When G61 (exact stop check mode) is selected.
- (b) When the G09 (exact stop check) is issued in the same block.
- (c) when the error detect switch (external signal) is ON.

(3) Deceleration check system

Deceleration check is a system that executes the next block only after the command deceleration check is executed as shown below, and it has been confirmed that the position error amount, including the servo system, is less than the in-position check width (designated with parameter or with "I" in same block).



12 Programming Support Functions

12.2 Machining Accuracy Support Functions

12.2.2.1 Exact Stop Mode

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
	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

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
	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 Detect

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
	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 Programming Support Functions

12.2 Machining Accuracy Support Functions

12.2.2.4 Programmable Inposition Check

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

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 Programming Support Functions

12.2 Machining Accuracy Support Functions

12.2.3 High-Accuracy Control (G61.1)

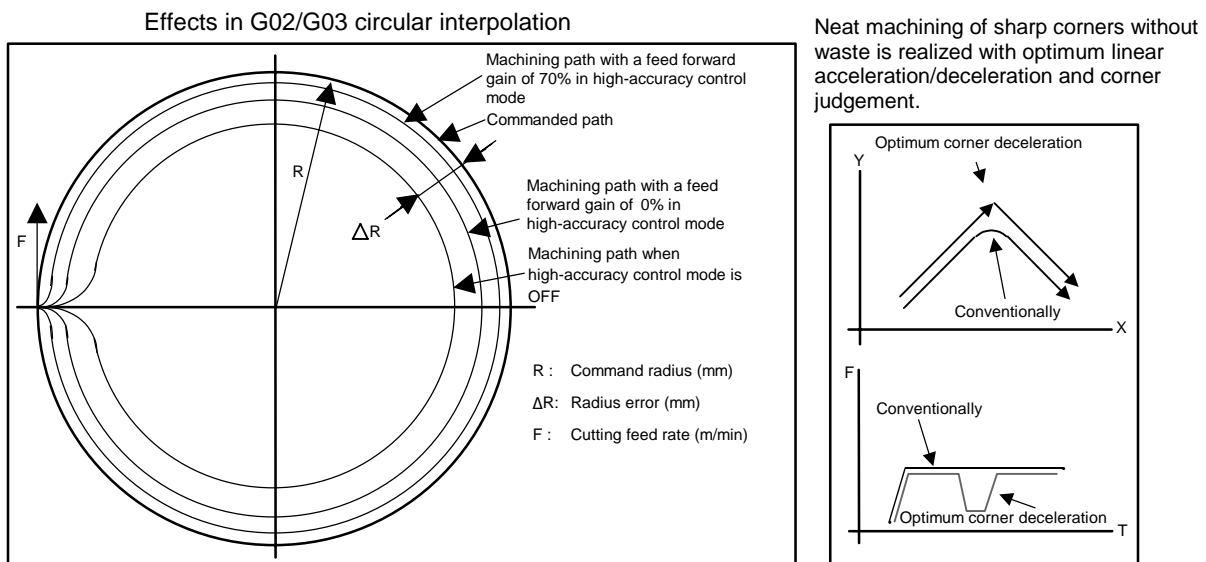
M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	-	-	-	-	-	-	-	-

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 function is a combination of the following element functions.

The high-accuracy control is commanded with ;

G61.1 High-accuracy control ON



(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.

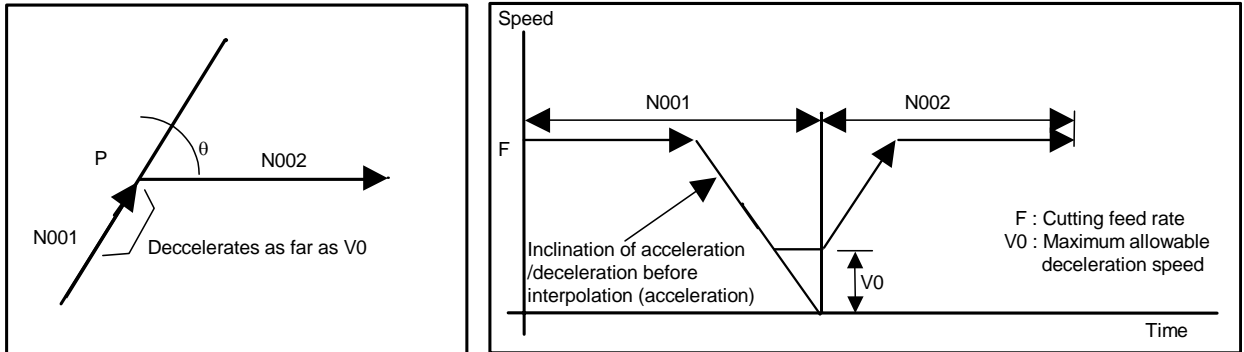
(Optimum corner deceleration is a function of high-accuracy control mode.)

12 Programming Support Functions

12.2 Machining Accuracy Support Functions

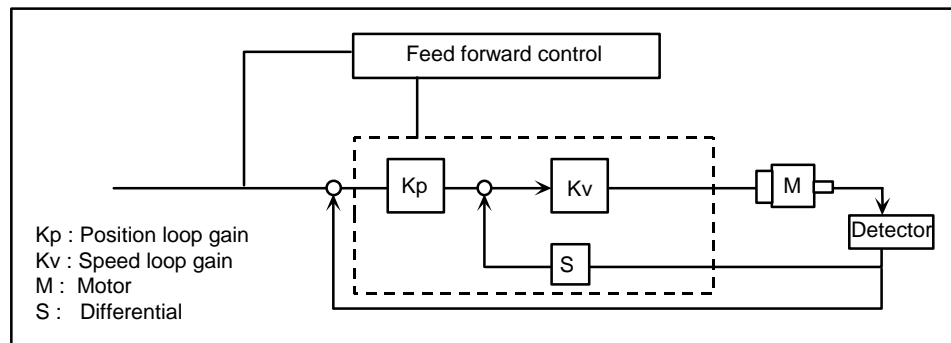
The speed change can be smoothed by the S-shape filter, the machine vibration can be suppressed, and the surface accuracy improved.

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.



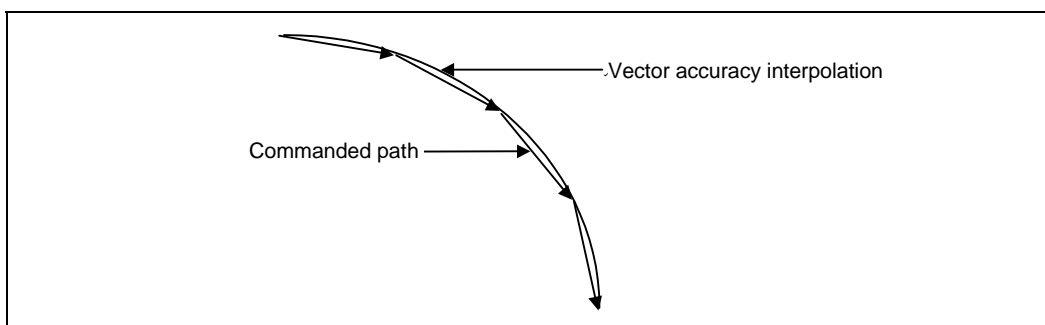
(3) Feed forward control

A stable servo control with an extremely small servo error can be realized using the feed forward control characteristic to this CNC system.



(4) Vector accuracy interpolation

When a fine segment is commanded and the angle between the blocks is extremely small (when not using optimum corner deceleration), interpolation can be carried out more smoothly using the vector accuracy interpolation.



12 Programming Support Functions

12.2 Machining Accuracy Support Functions

(5) Arc entrance/exit speed control

There are cases when the speed fluctuates and the machine vibrates at the joint from the straight line to arc or from the arc to straight line.

This function decelerates to the deceleration speed before entering the arc and after exiting the arc to reduce the machine vibration. If this is overlapped with corner deceleration, the function with the slower deceleration speed is valid.

(6) S-pattern filter control

This control interpolates while smoothing the changes in the segments distributed to each axis element with vector accuracy interpolation. With this, the fluctuation amplified by feed forward control is reduced and the effect onto the machine is reduced.

12 Programming Support Functions

12.3 Programming Support Functions

12.3 Programming Support Functions

12.3.1 Playback

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system	M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
	L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ

By repeatedly operating the controls on the panel, the amounts by which the machine is to move by jog feed, rapid traverse and handle feed can be converted into the command format of the control unit, and by repeatedly writing this data into the memory, machining programs for all the steps can be prepared.

12.3.2 Address Check

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system	M system	○	○	○	○	○	○	○	○
	L system	○	○	○	○	○	○	○	○

When a machining program is to be run, it can be checked in 1-word units. A parameter is used to select whether or not to conduct an address check.

Program address check operation

In addition to the conventional program check, a simple check in 1-word units is conducted. If letters of the alphabet follow successively, a program error results.

(Word: Consists of one letter followed by a number composed of several digits.)

With the conventional method, when a letter was not followed by a number, that the number was assumed to be zero, however, now an error will result when this new check is performed.

An error will not result in the following cases:

- (1) Machine language
- (2) Comment statements

Example of a program address check

Example 1: When the letter is not followed by a number

G28X; → Program should be reviewed and changed to G28X0; , etc.

Example 2: When there is an illegal character string

TEST; → Program should be reviewed and changed to "(TEST);", etc.

13 Machine Accuracy Compensation

13.1 Static Accuracy Compensation

13 Machine Accuracy Compensation

13.1 Static Accuracy Compensation

13.1.1 Backlash Compensation

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

This function compensates for the error (backlash) produced when the direction of the machine system is reversed.

The backlash compensation can be set in the cutting feed mode or rapid traverse mode.

The amount of backlash compensation can be set separately for each axis. It is set using a number of pulses in increments of one-half of the least 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 Machine Accuracy Compensation

13.1 Static Accuracy Compensation

13.1.2 Memory-type Pitch Error Compensation

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ

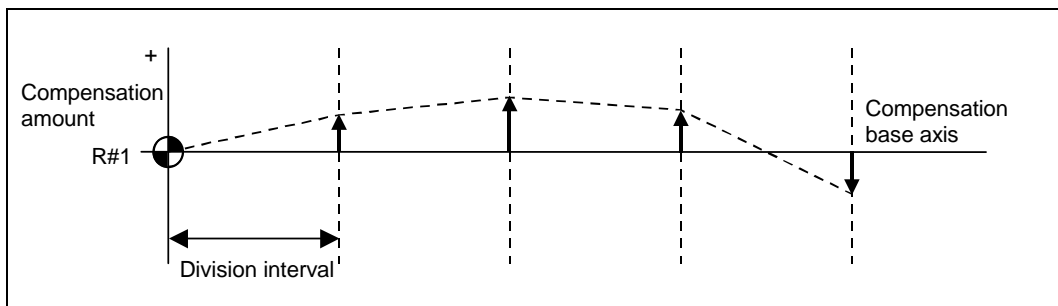
The machine accuracy can be improved by compensating for the errors in the screw pitch intervals among the mechanical errors (production errors, wear, etc.) of the feed screws.

The compensation positions and amounts are stored in the memory by setting them beforehand for each axis, and this means that there is no need to attach dogs to the machine.

The compensation points are divided into the desired equal intervals.

1. Division intervals of compensation points : 1 to 9999999 (output unit)
2. Number of compensation points : 1024
3. Compensation amount : -128 to 127 pulses (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 position serves as the zero (0) point. Thus, memory-type pitch error compensation is not performed if return to reference position 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 Machine Accuracy Compensation

13.1 Static Accuracy Compensation

13.1.3 Memory-type Relative Position Error Compensation

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ

Machine accuracy can be improved by compensating a relative error between machine axes, such as a production error or time aging.

The compensation base axis and compensation execution axis are set by using parameters.

The compensation points are divided at any desired equal intervals.

1. Compensation point dividing intervals : 1 to 9999999 (output unit)
 2. Number of compensation points : 1024
 3. Compensation amount : -128 to 127 pulses (detection unit)
 4. No. of compensated axes : 10 axes (including number of axes for memory type pitch error compensation.)
- (1) The compensation position is set for the compensation axis whose reference position serves as the zero (0) point. Thus, memory-type relative position error compensation is not performed if return to reference position 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 1,024 is a total including the points for memory-type pitch error compensation.

(Note 2) A scale of 0 to 99-fold is applied on the compensation amount.

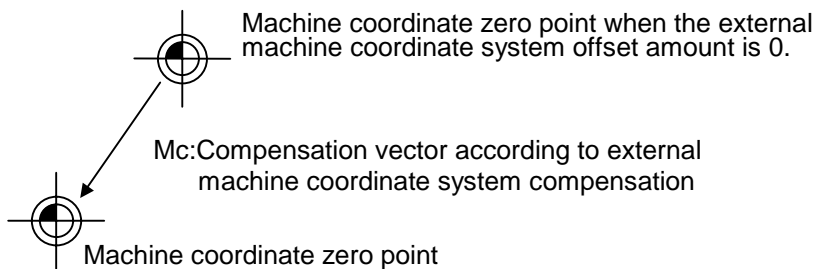
13 Machine Accuracy Compensation

13.1 Static Accuracy Compensation

13.1.4 External Machine Coordinate System Compensation

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ

The coordinate system can be shifted by inputting a compensation amount from the PLC. This compensation amount will not appear on the counter (all counters including machine position). If the machine's displacement value caused by heat is input for example, this can be used for thermal displacement compensation.



13.1.5 Circular Radius Error Compensation

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ

With commands designated during arc cutting, this function compensates for movement toward the inside of the arcs caused by a factor such as servo delay.

13 Machine Accuracy Compensation

13.2 Dynamic Accuracy Compensation

13.2 Dynamic Accuracy Compensation

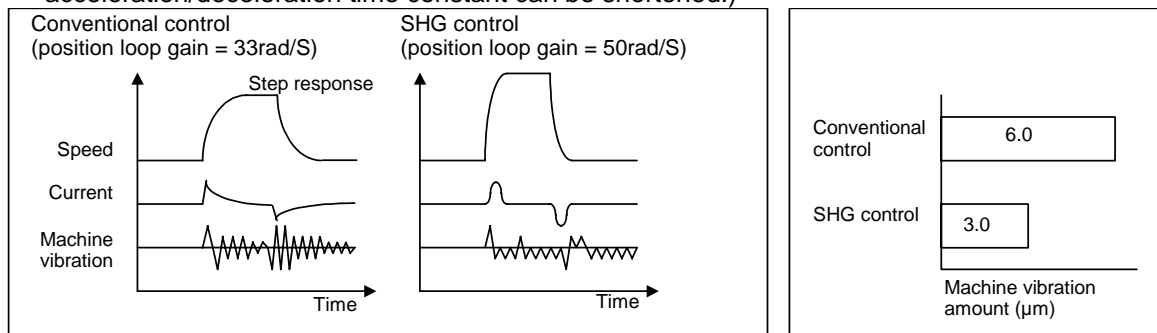
13.2.1 Smooth High-gain Control (SHG Control)

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ

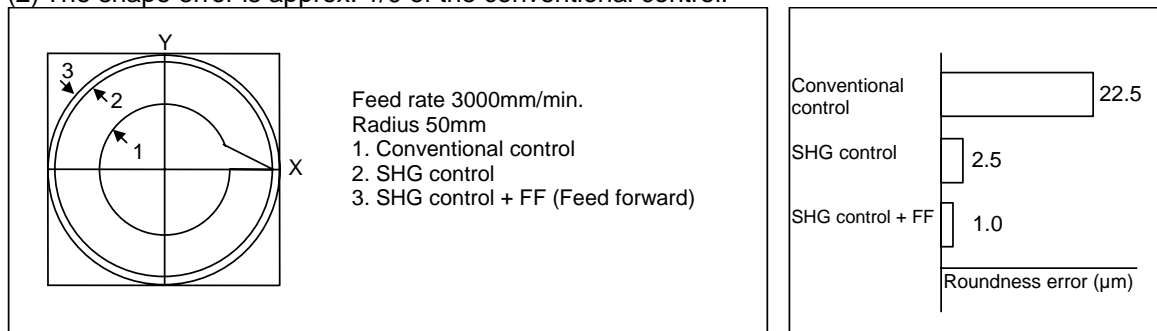
This is a high-response and stable position control method using the servo system (MDS- \square -V \square /SVJ2). This SHG control realizes an approximately three-fold position loop gain equally compared to the conventional control method.

The features of the SHG control are as follows.

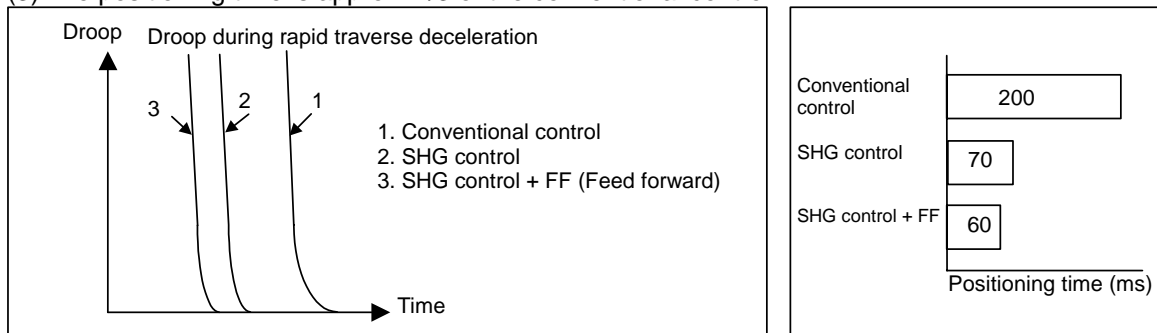
- (1) The acceleration/deceleration becomes smoother, and the mechanical vibration can be suppressed (approx. 1/2) during acceleration/deceleration. (In other words, the acceleration/deceleration time constant can be shortened.)



- (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 Machine Accuracy Compensation

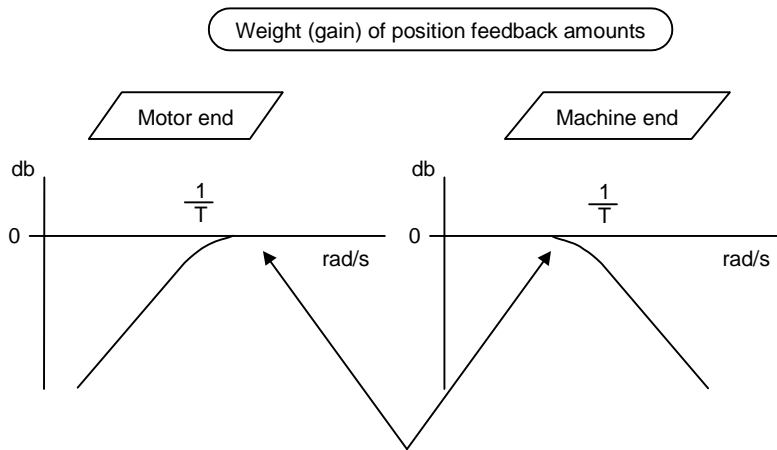
13.2 Dynamic Accuracy Compensation

13.2.2 Dual Feedback

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	△	△	△	△	△	△	△	△
L system	△	△	△	△	△	△	△	△

Depending on the frequency, the weight (gain) of the position feedback amount provided by the motor end detector and position feedback amount provided by the machine end detector stands in the correlation shown in the figure below. Semi-closed control is provided on a transient basis whereas positioning can be controlled by the closed status.

This function is used to select the primary delay filter time constant during dual feedback control as a parameter setting.



Time constant T here is adjusted using a parameter.

13.2.3 Lost Motion Compensation

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

This function compensates the error in the protrusion shape caused by lost motion at the arc quadrant changeover section during circular cutting.

14 Automation Support Functions

14.1 External Data Input

14 Automation Support Functions

14.1 External Data Input

By using the DDB interface, the following functions can be realized from the PLC ladder.

14.1.1 External Search

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ

The program No. and sequence No. to be automatically started in the memory or tape mode, ladder can be searched from the PLC ladder. The currently searched details can be read.

14 Automation Support Functions

14.1 External Data Input

14.1.2 External Workpiece Coordinate Offset

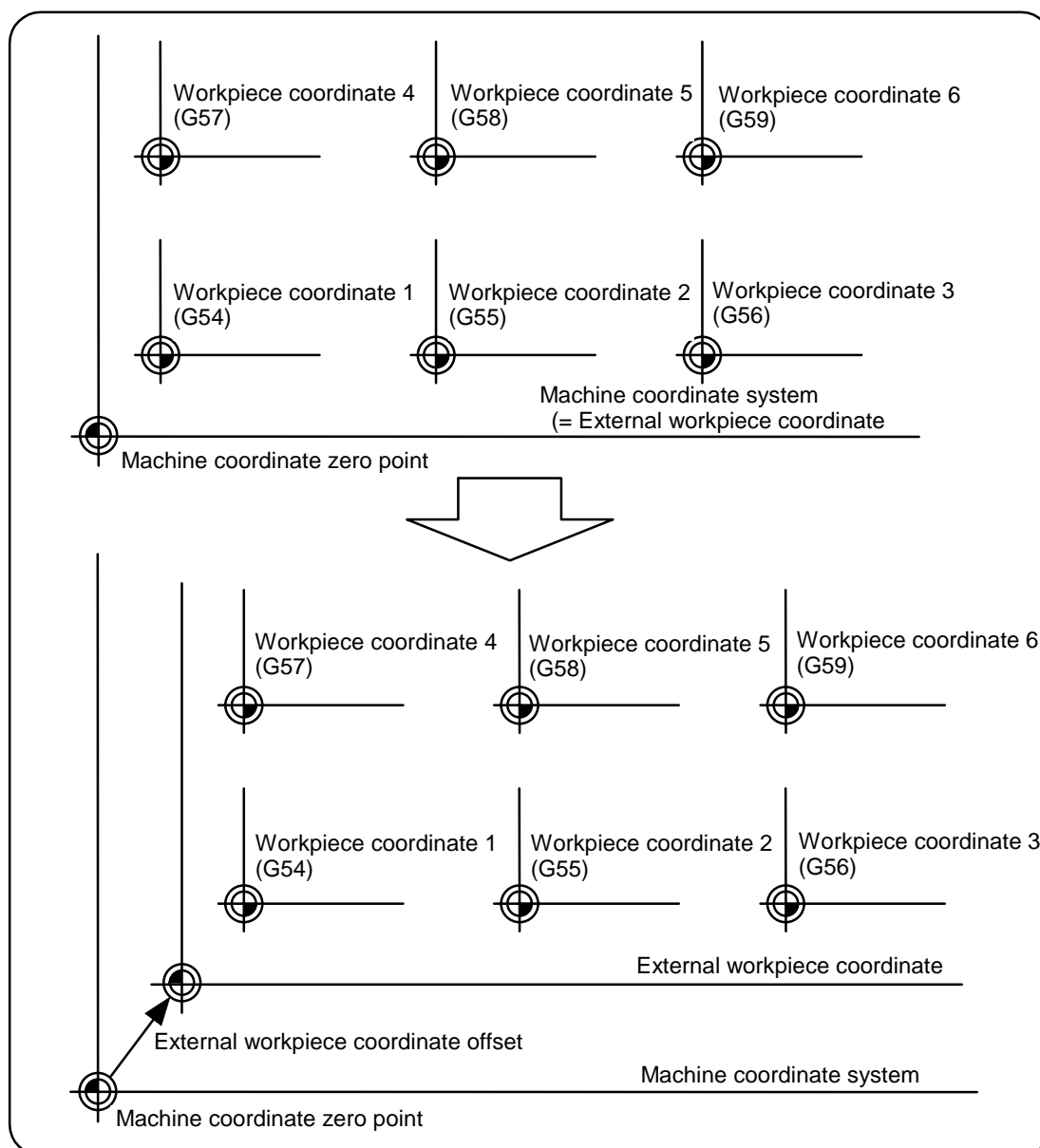
M/L system	Model	M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system		○	○	○	○	○	○	○	○
L system		○	○	○	○	○	○	○	○

External workpiece coordinate offset that serves as the reference for all the workpiece coordinate systems is available outside the workpiece coordinates.

By setting the external workpiece coordinate offset, the external workpiece coordinate system can be shifted from the machine coordinate system, and all the workpiece coordinate systems can be simultaneously shifted by an amount equivalent to the offset.

When the external workpiece coordinate offset is zero, the external workpiece coordinate systems coincide with the machine coordinate system.

It is not possible to assign movement commands by selecting the external workpiece coordinates.



14 Automation Support Functions

14.1 External Data Input

14.1.3 External Tool Offset

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	○	○	○	○
L system	-	-	-	-	○	○	○	○

The tool offset amount can be referred and updated from the PLC ladder.

14 Automation Support Functions

14.2 Measurement

14.2 Measurement

14.2.1 Skip

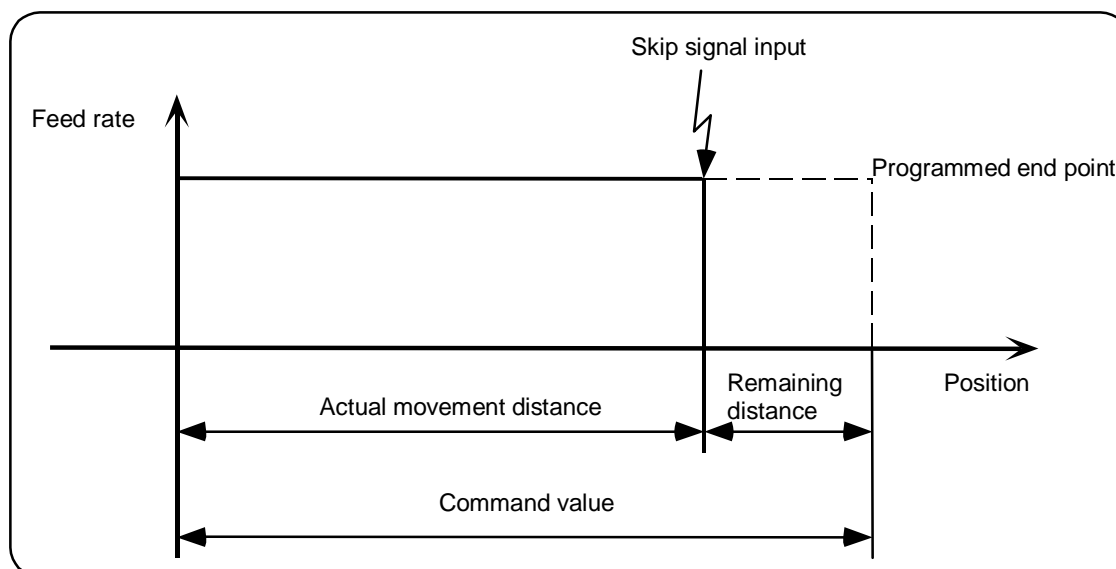
14.2.1.1 Skip

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ

When the external skip signal is input during linear interpolation with the G31 command, the machine feed is stopped immediately, the remaining distance is discarded and the commands in the next block are executed.

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.

$$\delta \doteq \frac{F}{60} \times (T_p + t)$$

δ : Coasting distance (mm)
 F : G31 rate (mm/min)
 T_p : Position loop time constant (s) = (position loop gain)⁻¹
 T : Response delay time of 0.0035 (s)

(Note 2) Skipping during machine lock is not valid.

14 Automation Support Functions

14.2 Measurement

14.2.1.2 Multiple-step Skip

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ

(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 and 3 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 7. 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 7) 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 127. If it exceeds the specified range, a program error occurs.

14 Automation Support Functions

14.2 Measurement

Table 1 Valid skip signals

Skip signal command P	Valid skip signal						
	7	6	5	4	3	2	1
1							○
2						○	
3						○	○
4					○		
5					○		○
6					○	○	
7					○	○	○
8				○			
:	:	:	:	:	:	:	:
125	○	○	○	○	○		○
126	○	○	○	○	○	○	
127	○	○	○	○	○	○	○

14.2.1.4 PLC Skip

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	–	–	–	–	Δ	Δ	Δ	Δ
L system	–	–	–	–	Δ	Δ	Δ	Δ

This function enables skip operations to be performed by signals which are input from the user PLC.

14 Automation Support Functions

14.2 Measurement

14.2.1.5 Speed Change Skip

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	-	Δ	Δ
L system	-	-	-	-	-	-	Δ	Δ

This function is used to change the feed rate or to stop the movement by inputting the skip signal during the linear interpolation.

After stopping the movement with skipping, the remaining movement command is canceled and the next block is executed.

G31 Xx Yy Zz Ff F1 = f1 ... Fn = fn ;

(n is the skip signal 1 to 8.)

Xx, Yy, Zz : Command value

Ff : Feed rate when starting the cutting feed (mm/min)

fn : Feed rate after detecting the skip signal (mm/min)

fn=0: Movement stop

fn≠0: Changing the feed rate to fn

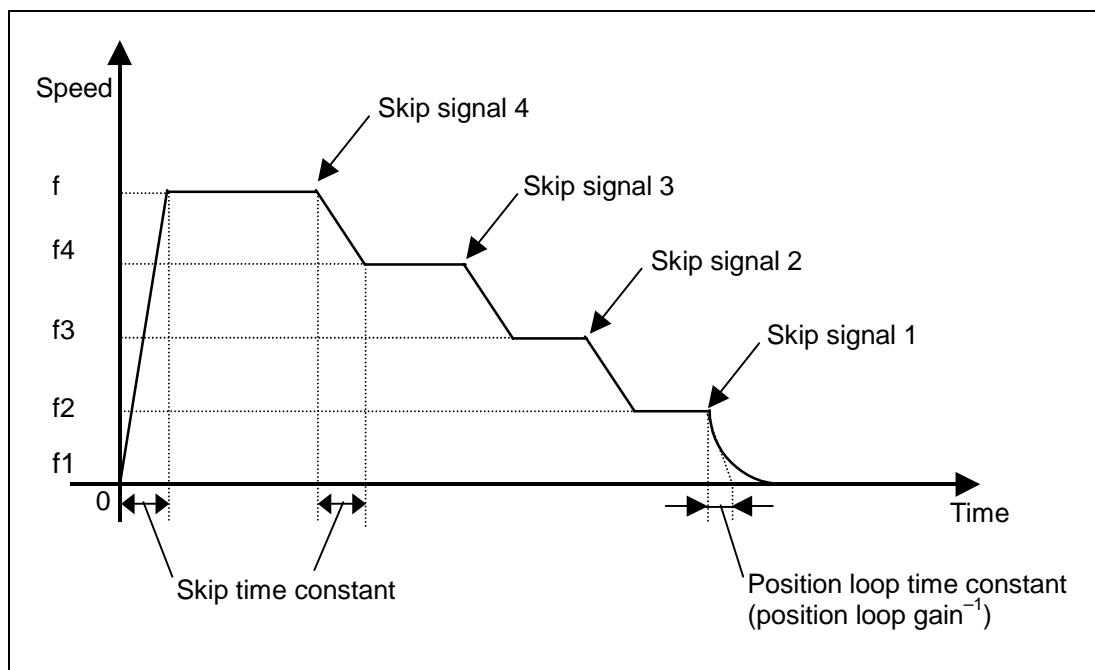
F1=Feed rate after inputting the skip signal 1

⋮

F8=Feed rate after inputting the skip signal 8

Operation example

G31 X100. Ff F1=0 F2 = f2 F3 = f3 F4 = f4;



14 Automation Support Functions

14.2 Measurement

14.2.5 Automatic Tool Length Measurement

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	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_ ;

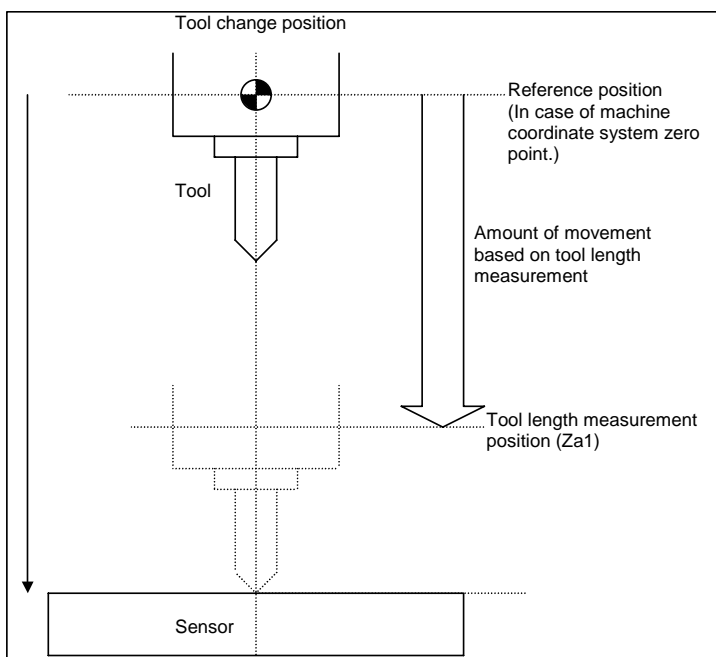
Z : Measurement axis address and measurement position coordinate. ... X, Y, Z, α (where α is an optional axis)

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.

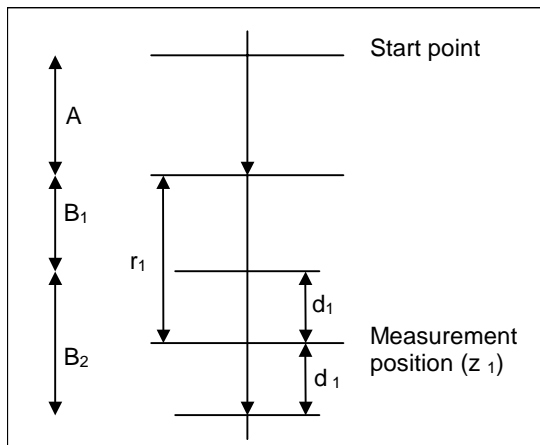
Example of program

```
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.

14 Automation Support Functions

14.2 Measurement



Area A : Moves with rapid traverse feed rate.

Areas B₁, B₂: Moves with the measurement speed (f₁ or parameter setting)

If a sensor signal is input in area B₁, an error will occur.

If a sensor signal is not input in the area B₂, an error will occur.

(2) Automatic tool length measurement (L series)

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.

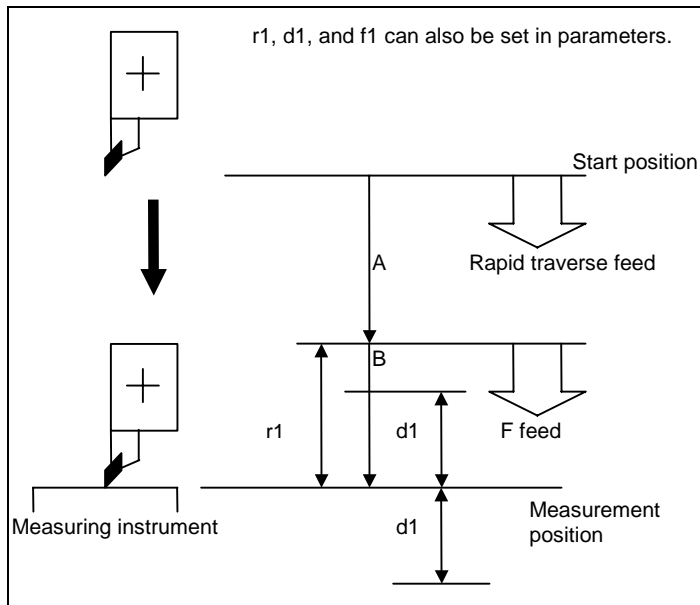
G37 α _R_D_F_ ;

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.

14 Automation Support Functions

14.2 Measurement



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 turns ON during the tool is moving in the B area, an error occurs. If the measurement position arrival signal does not turn ON although the tool passes through the measurement position x1 (z1) and moves d1, an error occurs.

14 Automation Support Functions

14.2 Measurement

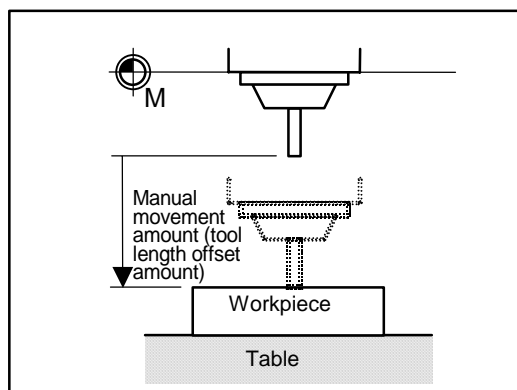
14.2.6 Manual Tool Length Measurement 1

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	△	△	△	△	△	△	△	△
L system	△	△	△	△	△	△	△	△

Simple measurement of the tool length is done without a sensor.

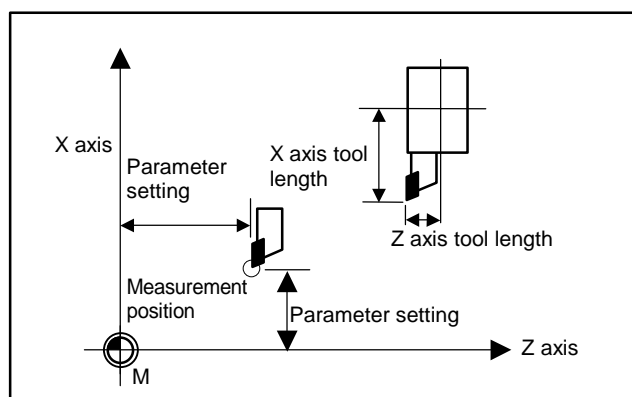
(1) Manual tool length measurement I [M system]

When the tool is at the reference position, this function enables the distance from the tool tip to the measurement position (top of workpiece) to be measured and registered as the tool length 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 Automation Support Functions

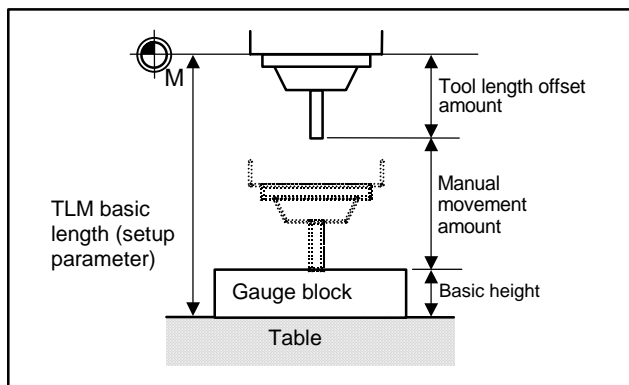
14.2 Measurement

14.2.7 Manual Tool Length Measurement 2

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	Δ	Δ	Δ	Δ
L system	-	-	-	-	Δ	Δ	Δ	Δ

(1) Manual tool length measurement II [M system]

When the tool is positioned at the reference position, this function enables the distance from the reference position to the tool tip to be measured and registered as the tool length offset amount. In this case, the position of the gauge block used as a reference must be set as the basic height.

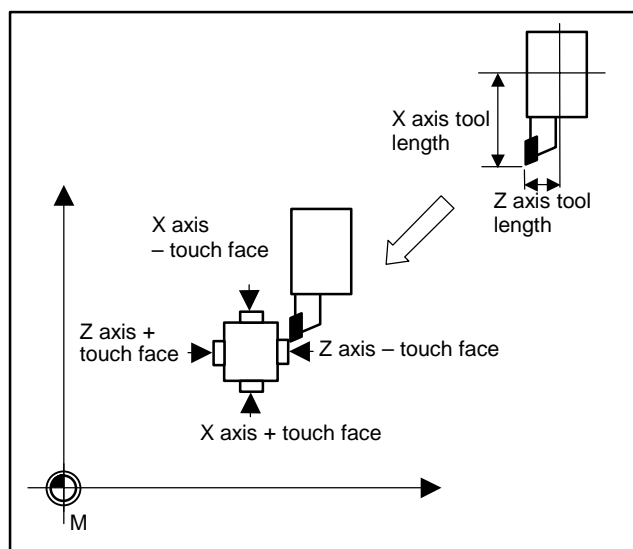


If the height axis designation parameter is ON, the axis designated for plane selection basic axis K is the axis targeted for measurement as the height axis.

Furthermore, if the tool length measurement check parameter is ON, an input OK/cancel confirmation message appears after input key has been pressed.

(2) Manual tool length measurement II [L system]

A device in which a touch sensor is built is used. Simply by causing the tool nose to touch the touch sensor in manual feed, the tool offset amount can be calculated and stored in tool offset amount memory.



Measurement method

- Preset the machine coordinates of the touch sensor touch face in parameter as the measurement basic value.
- Select the tool whose tool length offset amount is to be measured.
- Cause the tool nose to touch the touch sensor in manual feed.

The tool length offset amount is automatically calculated from the machine coordinates when the tool nose touches the touch sensor and the measurement basic value. It is stored in memory as tool length offset amount.

$$\begin{aligned} \text{Tool length offset amount} \\ = \text{machine coordinates} - \text{measurement basic value (sensor position)} \end{aligned}$$

14 Automation Support Functions

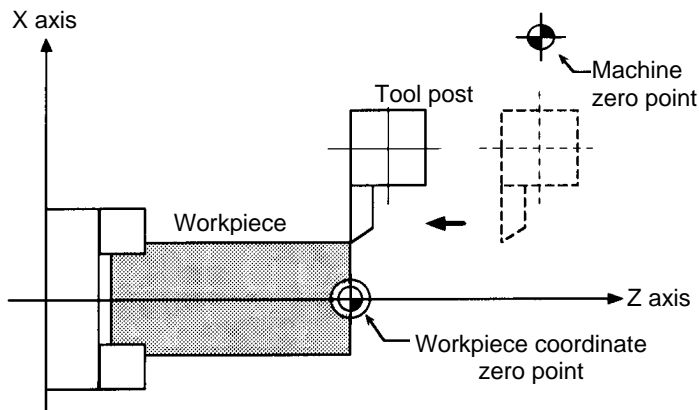
14.2 Measurement

14.2.8 Workpiece Coordinate Offset Measurement

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	Δ (*)	Δ (*)	Δ (*)	Δ (*)
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ

The external workpiece coordinate offset data for the Z axis can be set by cutting the workpiece face by means of manual operations and inputting the workpiece measurement signal.

Measurement of external workpiece coordinate offset data for Z axis



Setting method

- (1) Select the tool, and cut the workpiece face.
- (2) When the workpiece measurement signal is input, the external workpiece coordinate offset data for the Z axis is calculated from the machine coordinate values, length of the tool used and tool nose wear offset amount, and stored in the memory.

(Note) The (*) mark is the simple workpiece coordinate offset input. With this function, the workpiece coordinate system offset data automatic calculation value and machine position for corresponding axes are displayed in the setting part on the workpiece coordinate offset screen.

14 Automation Support Functions

14.2 Measurement

14.2.9 Workpiece Position Measurement

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	Δ	Δ	Δ
L system	-	-	-	-	-	-	-	-

The workpiece position measurement function is used to measure each axis' coordinate point by installing a sensor on the spindle and the sensor contacting the workpiece with the manual feed or handle feed.

The surface, hole center and width center coordinates are calculated from the measured coordinates, and those calculated results are set in the workpiece coordinate offset.

Only 1st part system is available for the workpiece position measurement.

(1) Surface workpiece offset measurement

The workpiece position measurement coordinates are calculated from the skip machine position of the X, Y and Z axes.

Measurement position coordinate X = X axis' skip machine position + Sensor diameter/2
+ Center compensation amount (Horizontal)

Measurement position coordinate Y = Y axis' skip machine position + Sensor diameter/2
+ Center compensation amount (Vertical)

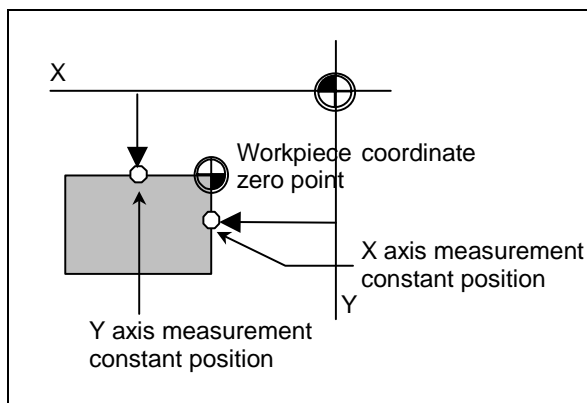
Measurement position coordinate Z = Z axis' skip machine position - Sensor length

The sensor diameter/2 changes between +/- with the tool movement direction during the measurement.

The sensor diameter and center compensation amount are applied to the X axis or Y axis.

The sensor length is applied to the Z axis.

The measurement position coordinate of the X axis, Y axis or Z axis is set in the specified workpiece coordinate offset.



To set the workpiece coordinate offset, the X axis is measured and the X axis' offset coordinate is set. Then, the Y axis' offset is measured and set. Finally, the Z axis' offset is measured and set.

14 Automation Support Functions

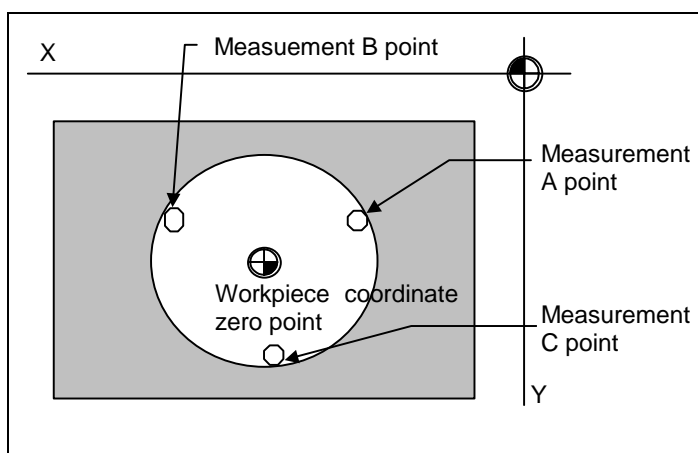
14.2 Measurement

(2) Hole center workpiece offset measurement

The measurement position coordinates of two axes (X, Y) are measured at three points, and the hole center is calculated. The calculated result is set in the specified workpiece coordinate offset. The workpiece position measurement coordinates are calculated from the skip machine position of the X, Y and Z axes.

Measurement position coordinate X = X axis' skip machine position + Center compensation amount
(Horizontal)

Measurement position coordinate Y = Y axis' skip machine position + Center compensation amount
(Vertical)



To set the workpiece coordinate offset, the position X and Y of the measurement A point are measured, and the measured values are set in the measurement A point. In the same manner as the measurement A point, the measurement B point and then C point are measured and set. The hole center coordinate is calculated by setting the workpiece coordinate system after setting three points, and the calculated result is set in the workpiece coordinate offset.

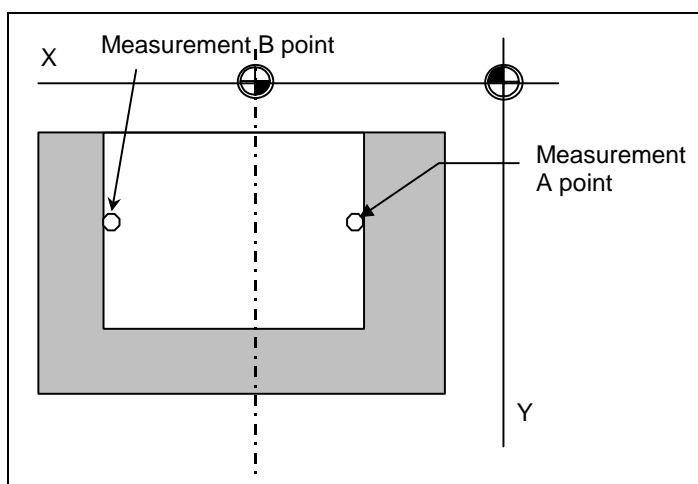
(3) Width center workpiece offset measurement

The measurement position coordinate of the X axis or Y axis is measured at two points, and each axis' groove center is calculated. The calculated result is set in the specified workpiece coordinate offset.

The workpiece position measurement coordinates are calculated from the skip machine position of the X, Y and Z axes.

Measurement position coordinate X = X axis' skip machine position + Center compensation amount
(Horizontal)

Measurement position coordinate Y = Y axis' skip machine position + Center compensation amount
(Vertical)



To set the workpiece coordinate offset, the position X (position Y) of the measurement A point is measured, and the measured value is set in the measurement A point. In the same manner as the measurement A point, the measurement B point is measured and set. The groove width center coordinate of the X axis (Y axis) is calculated by setting the workpiece coordinate system after setting two points, and the calculated result is set in the workpiece coordinate offset.

14 Automation Support Functions

14.3 Monitoring

14.3 Monitoring

14.3.1 Tool Life Management

Tool Life Management I

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	Δ	Δ	Δ	Δ
L system	-	-	-	-	Δ	Δ	Δ	Δ

(1) M series

The time (0 to 4000 hours) and frequency (0 to 65000 times) of use of the user PLC specified tool are accumulated. Tool data including the time and frequency of use of the PLC specified tool is output.

(2) L series

Tool life management is performed using the time and frequency of use of a tool.

(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 number of the specified tool is incremented each time a tool selection (T) command is specified for the tool.

If the counter reaches the life time when a tool selection command is specified, an alarm is given.

Tool Life Management II

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ

(1) M series

A spare tool change function is added to tool life management I. This function selects a usable tool out of the spare tools of the group determined by the value specified by the user PLC, then outputs data of such usable spare tool. The spare tool can be selected in two ways: the tools are selected in order they were registered in the group or the tool whose remaining life is the longest of all in the group is selected.

(2) L series

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 Automation Support Functions

14.3 Monitoring

14.3.2 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 systems according to the model.)

20/40/80 sets

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	-	-	-
L system	Δ80	Δ80	Δ80	Δ80	Δ80	Δ80	Δ80	Δ80

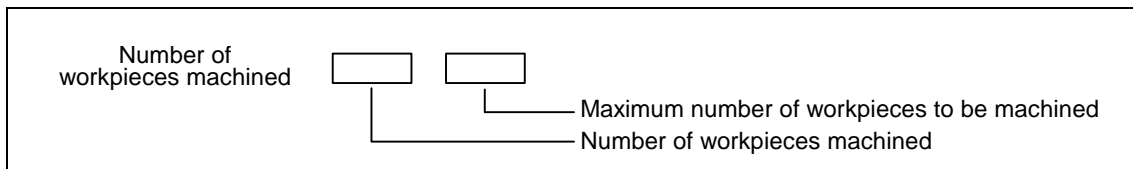
100/200 sets

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ100	Δ100	Δ100	Δ100	Δ200	Δ200	Δ200	Δ200
L system	-	-	-	-	-	-	-	-

14.3.3 Display of Integrated Time/Number of Parts

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

The number of machined parts is counted up each time a part is machined, and displayed .



14.3.4 Load Meter

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

Using the user PLC, this function displays the spindle load, Z-axis load, etc. in the form of bar graphs.

14 Automation Support Functions

14.3 Monitoring

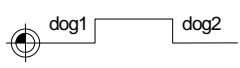

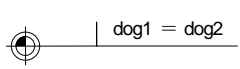
14.3.5 Position Switch

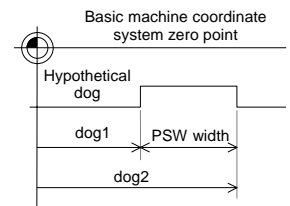
M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○8	○8	○8	○8	○24	○24	○24	○24
L system	○8	○8	○8	○8	○24	○24	○24	○24

The position switch (PSW) function provides hypothetical dog switches in place of the dog switches provided on the machine axes by setting the axis names and coordinates indicating the hypothetical dog positions as parameters beforehand so that signals are output to the PLC interface when the machine has reached these hypothetical dog positions. The hypothetical dog switches are known as position switches (PSW).

The coordinates 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 (M60: PSW1 to PSW8, M60S: PSW1 to PSW24). 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



14 Automation Support Functions

14.5 Others

14.5 Others

14.5.1 Programmable Current Limitation

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

This function allows the current limit value of the servo axis to be changed to a desired value in the program, and is used for the workpiece stopper, etc.

The commanded current limit value is designated with a ratio of the limit current to the rated current.

The current limit value can also be set from the D.D.B. 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 Safety and Maintenance

15.1 Safety Switches

15.1.1 Emergency Stop

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		○	○	○	○	○	○	○	○
	M system	○	○	○	○	○	○	○	○
	L system	○	○	○	○	○	○	○	○

All operations are stopped by the emergency stop signal input and, at the same time, the drive section is stopped using the dynamic brake and the movement of the machine is stopped. At this time, the READY lamp on the setting and display unit goes OFF and the servo ready signal is turned OFF.

15.1.2 Data Protection Key

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		○	○	○	○	○	○	○	○
	M system	○	○	○	○	○	○	○	○
	L system	○	○	○	○	○	○	○	○

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 and common variables

Group 3: For protecting the machining programs

15 Safety and Maintenance

15.2 Display for Ensuring Safety

15.2 Display for Ensuring Safety

15.2.1 NC Warning

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

The warnings which are output by the NC system are listed below. When one of these warnings has occurred, a warning number is output to the PLC and a description of the warning appears on the screen. Operation can be continued without taking further action.

Type of warning	Description
Servo warning	The servo warning is displayed.
Spindle warning	The spindle warning is displayed.
System warning	The system warning is displayed. (State such as temperature rise, battery voltage low, etc.)
Absolute position warning	A warning in the absolute position detection system is displayed.
Auxiliary axis warning	The auxiliary axis warning is displayed.

15.2.2 NC Alarm

The alarms which are output by the NC system are listed below. When one of these alarms has occurred, an alarm number is output to the PLC, and a description of the alarm appears on the screen. Operation cannot be continued without taking remedial action.

Type of 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.
Auxiliary axis alarm	The auxiliary axis alarm is displayed.
Computer link error	The computer link alarm is displayed.
User PLC alarm	The user PLC alarm is displayed.
Program error	This alarm occur during automatic operation, and the cause of this alarm is mainly program errors which occur, for instance, when mistakes have been made in the preparation of the machining programs or when programs which conform to the specification have not been prepared.

15 Safety and Maintenance

15.2 Display for Ensuring Safety

15.2.3 Operation Stop Cause

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

The stop cause of automatic operation is displayed on the setting and display unit.

15.2.4 Emergency Stop Cause

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

When "EMG" (emergency stop) message is displayed in the operation status display area of the setting and display unit, the emergency stop cause can be confirmed.

15 Safety and Maintenance

15.2 Display for Ensuring Safety

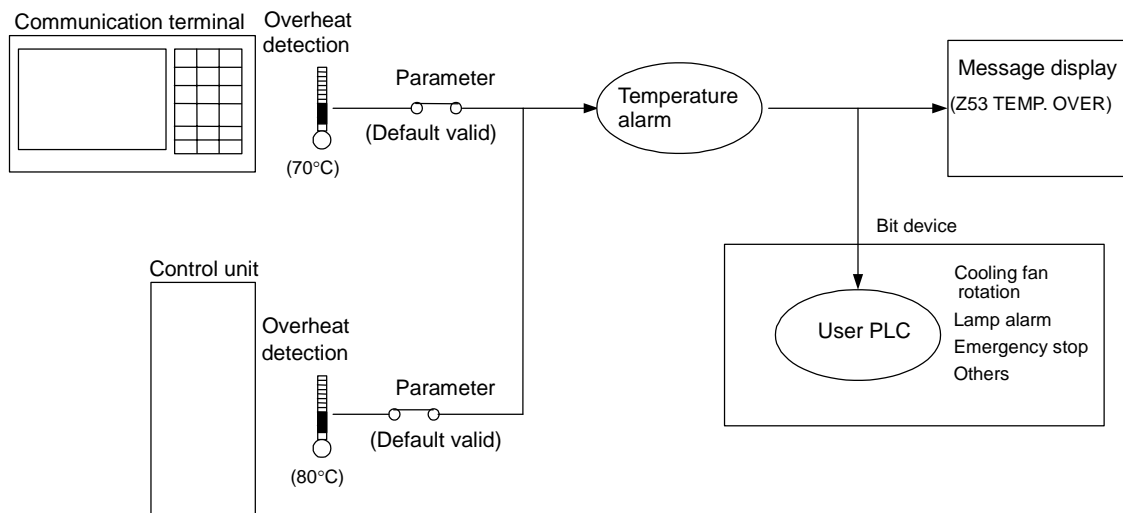
15.2.5 Temperature Detection

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	○	○	○	○
L system	-	-	-	-	○	○	○	○

When overheating is detected in the control unit or the communication terminal, an overheat signal is output at the same time as the alarm is displayed. 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 overheat signal is turned OFF.

The overheat alarm occurs at 80°C or more for the control unit or 70°C or more for the communication terminal.



(Note 1) If the parameter is used to set the temperature rise detection function to invalid, overheating may occur, thereby disabling control and possibly resulting in the axes running out of control, which in turn may result in machine damage and/or bodily injury or destruction of the unit. It is for this reason that the detection function is normally left "valid" for operation.

15 Safety and Maintenance

15.3 Protection

15.3 Protection

15.3.1 Stroke End (Over Travel)

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

When limit switches and dogs have been attached to the machine and a limit switch has kicked a dog, the movement of the machine is stopped by the signal input from the limit switch.

At the same time, the alarm output is sent to the machine.

The stroke end state is maintained and the alarm state is released by feeding the machine in the reverse direction in the manual mode to disengage the dog.

15.3.2 Stored Stroke Limit

The stored stroke limits I, II, IIB, IB and IC are handled as follows.

Type	Prohibited range	Explanation
I	Outside	<ul style="list-style-type: none"> •Set by the machine maker. •When used with II, the narrow range designated by the two types becomes the movement valid range.
II	Outside	<ul style="list-style-type: none"> •Set by the user.
IIB	Inside	<ul style="list-style-type: none"> •The change or function of parameter can be turned OFF/ON with the program command. •Select II or IIB with the parameters.
IB	Inside	<ul style="list-style-type: none"> •Set by the machine maker.
IC	Outside	<ul style="list-style-type: none"> •Set by the machine maker. •Can be rewritten with DDB.

15 Safety and Maintenance

15.3 Protection

15.3.2.1 Stored Stroke Limit I/II

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

(1) Stored Stroke Limit I

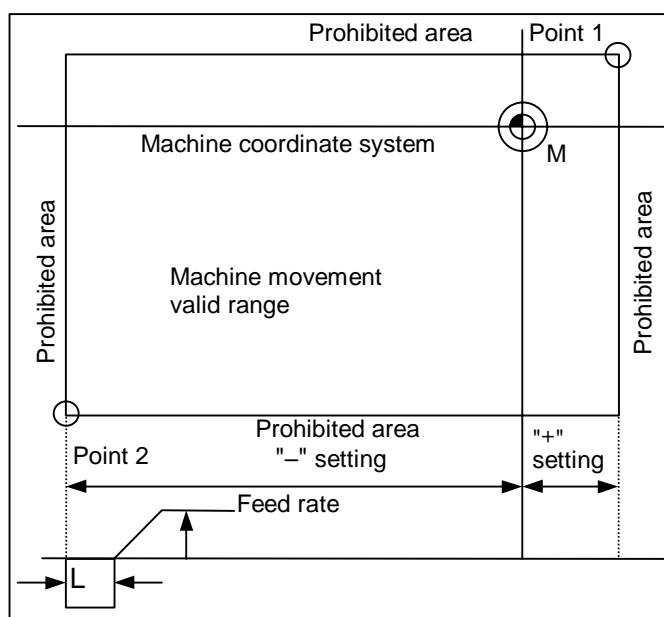
This is the stroke limit function used by the machine maker, and the area outside the set limits is the entrance prohibited area.

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 position 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.

15 Safety and Maintenance

15.3 Protection

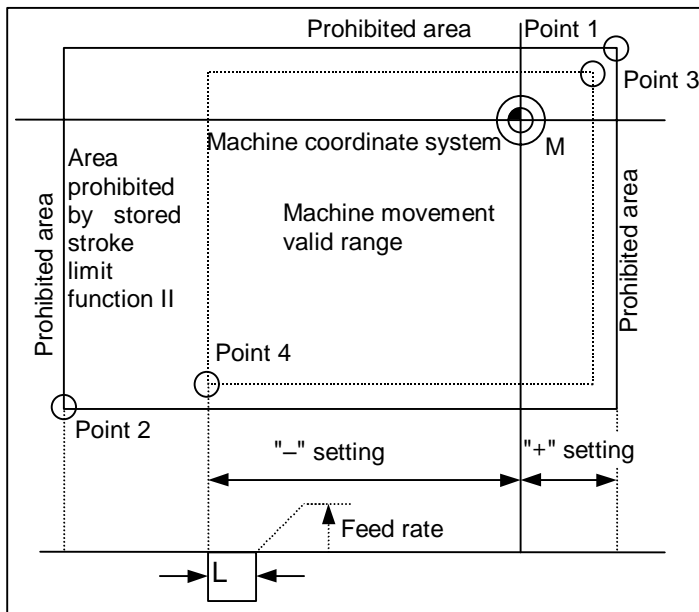
(2) Stored Stroke Limit II

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 $+99999.999$ mm.

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.

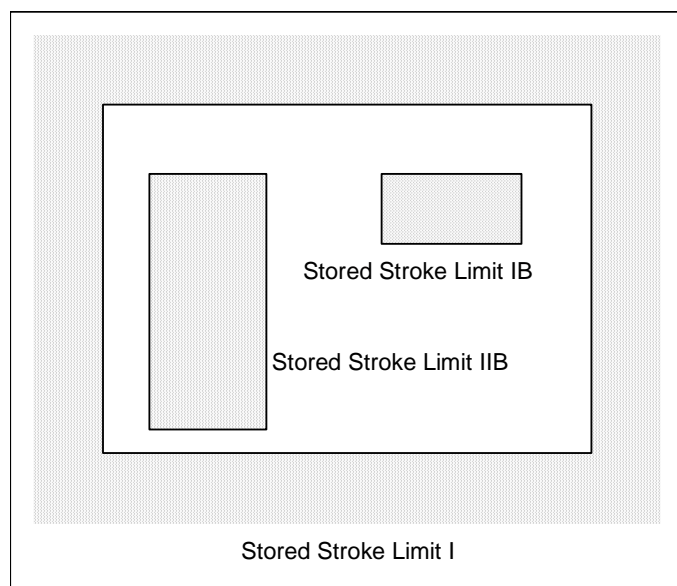
15 Safety and Maintenance

15.3 Protection

15.3.2.2 Stored Stroke Limit IB

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	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.



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 Safety and Maintenance

15.3 Protection

15.3.2.3 Stored Stroke Limit IIB

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ

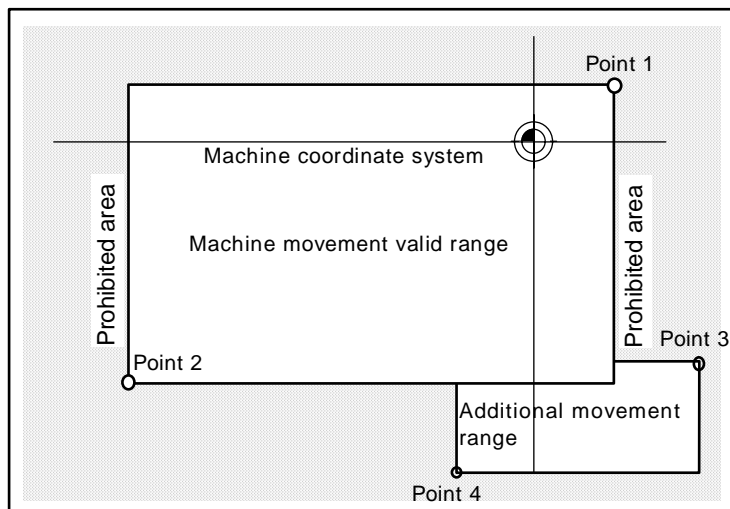
A parameter is used to switch between this function and stored stroke limit II. With stored stroke limit IIB, the range inside the boundaries which have been set serves as the tool entry prohibited area.

15.3.2.4 Stored Stroke Limit IC

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	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 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.

15 Safety and Maintenance

15.3 Protection

15.3.4 Chuck/Tail Stock Barrier Check

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	-	-	-
L system	○	○	○	○	○	○	○	○

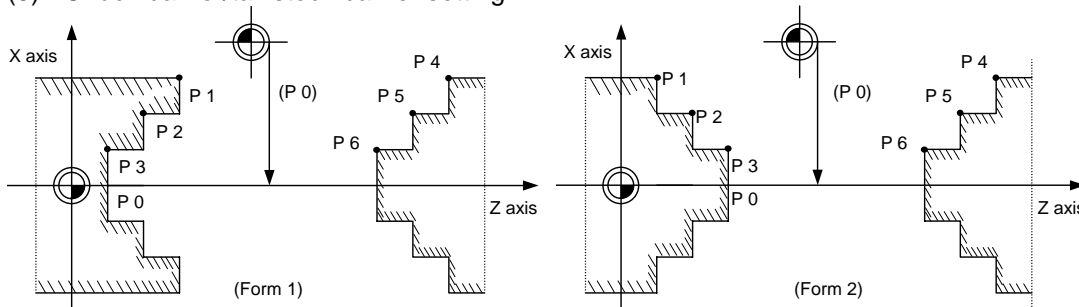
By limiting the tool nose point move range, this function prevents the tool from colliding with the chuck or tail stock because of a programming error.

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 position 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:

$$P1 \geq P2 \geq P3, P4 \geq P5 \geq P6$$

(However, this need not apply to the Z axis coordinates.)

15 Safety and Maintenance

15.3 Protection

15.3.5 Interlock

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

The machine movement will decelerate and stop as soon as the interlock signal, serving as the external input, is turned ON.

When the interlock signal is turned OFF, the machine starts moving again.

- (1) In the manual mode, only that axis for which the interlock signal is input will stop.
- (2) In the automatic mode, all axes will stop when the interlock signal is input to even one axis which coincides with the moving axis.
- (3) Block start interlock

While the block start interlock signal (*BSL) is OFF (valid), the execution of the next block during automatic operation will not be started. The block whose execution has already commenced is executed until its end. Automatic operation is not suspended. The commands in the next block are placed on standby, and their execution is started as soon as the signal is turned ON.

(Note 1) This signal is valid for all blocks including internal operation blocks such as fixed cycles.

(Note 2) This signal (*BSL) is set ON (invalid) when the power is turned ON. If it is not used, there is no need to make a program with the PLC.

- (4) Cutting start interlock

While the cutting start interlock signal (*CSL) is OFF (valid), the execution of all movement command blocks except positioning during automatic operation will not be started. The block whose execution has already commenced is executed until its end. Automatic operation is not suspended. The commands in the next block are placed on standby, and their execution is started as soon as the signal is turned ON.

(Note 1) The signal is valid for all blocks including internal operation block such as fixed cycles.

(Note 2) This signal (*CSL) is set ON (invalid) when the power is turned ON. If it is not used, there is no need to make a program with the PLC.

15.3.6 External Deceleration

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

This function reduces the feed rate to the deceleration speed set by the parameter when the external deceleration input signal, which is the external input from the user PLC, has been set to ON. External deceleration input signals are provided for each axis and for each movement direction ("+" and "-"), and a signal is valid when the signal in the direction coinciding with the direction of the current movement has been input. When an axis is to be returned in the opposite direction, its speed is returned immediately to the regular speed assigned by the command.

When non-interpolation positioning is performed during manual operation or automatic operation, only the axis for which the signal that coincides with the direction of the current movement has been input will decelerate.

However, with interpolation during automatic operation, the feed rate of the axis will be reduced to the deceleration rate if there is even one axis for which the signal that coincides with the direction of current movement has been input.

The external deceleration input signal can be canceled using a parameter for the cutting feed only.

15 Safety and Maintenance

15.3 Protection

15.3.8 Door Interlock

15.3.8.1 Door Interlock I

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		○	○	○	○	○	○	○	○
M system		○	○	○	○	○	○	○	○
L system		○	○	○	○	○	○	○	○

Outline of function

Under the CE marking scheme of the European safety standards (machine directive), the opening of any protection doors while a machine is actually moving is prohibited.

When the door open signal is input from the PLC, this function first decelerates and stops all the control axes, establishes the ready OFF status, and then shuts off the drive power inside the servo amplifiers so that the motors are no longer driven.

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: this can be done either by operations performed by the operator or by using the user PLC.

15 Safety and Maintenance

15.3 Protection

15.3.8.2 Door Interlock II

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

Outline of function

Under the CE marking scheme of the European safety standards (machine directive), the opening of any protection doors while a machine is actually moving is prohibited.

When the door open signal is input from the PLC, this function first decelerates and stops all the control axes, establishes the ready OFF status, and then shuts off the drive power inside the servo amplifiers so that the motors are no longer driven.

With the door interlock function established by the door open II signal, automatic start can be enabled even when the door open signal has been input. However, the axes will be set to the interlock status.

Description of operation

When a door is open

The NC system operates as follows when the door open II signal is input:

- (1) It stops operations.
All the axes decelerate and stop.
The spindle also stops.
- (2) The complete standby status is established.
- (3) After all the 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: this can be done either by operations performed by the operator or by using the user PLC.

(Note) Concerning the handling of an analog spindle

The signals described in this section are valid in a system with bus connections for the NC control unit and drive units. When an analog spindle is connected, the NC system cannot verify that the spindle has come to a complete stop. This means that the door should be opened after the PLC has verified that the spindle has come to a complete stop. Since the spindle may resume its rotation immediately after the door has been closed, set the forward and reverse rotation signals to OFF when opening the door so as to ensure safety.

15 Safety and Maintenance

15.3 Protection

15.3.11 Program Display Lock

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

This function allows the display of only a target program (label address 9000) to be invalidated for the program display in the monitor screen, etc.

The operation search of a target program can also be invalidated.

The validity of the display is selected with the parameters. The setting will be handled as follows according to the value.

0: Display and search are possible.

1: Display of the program details is prohibited.

2: Display and operation search of the program details are prohibited.

The program details are not displayed in the prohibited state, but the program number and sequence number will be displayed.

15 Safety and Maintenance

15.4 Maintenance and Troubleshooting

15.4 Maintenance and Troubleshooting

15.4.1 History Diagnosis

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
	M system	○	○	○	○	○	○	○	○
	L system	○	○	○	○	○	○	○	○

This is a maintenance function which is useful for tracing down the history and NC operation information and analyzing trouble, etc. This information 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.

(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 through the RS-232C interface.

15.4.2 Setup / Monitor for Servo and Spindle

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
	M system	○ monitor	○ monitor	○ monitor	○ monitor	○ monitor	○ monitor	○ monitor	○ monitor
	L system	○ monitor	○ monitor	○ monitor	○ monitor	○ monitor	○ monitor	○ monitor	○ monitor

The information on the servos (NC axes), spindles, PLC axes and power supplies appears on the setting and display unit.

Main information displayed on the monitor:

Position loop tracking deviation, motor speeds, load current, detector feedback, absolute position detection information, drive unit alarm histories, operation times, drive unit software versions, etc.

15.4.3 Data Sampling

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
	M system	○	○	○	○	○	○	○	○
	L system	○	○	○	○	○	○	○	○

Sampling of the servo and spindle data for which an alarm occurrence is a stop condition is performed all the time. By using the waveform display function, this sampling data can be displayed in the waveforms.

The data currently displayed can be stored on a memory card, and read out when required.

15 Safety and Maintenance

15.4 Maintenance and Troubleshooting

15.4.4 Waveform Display

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	-	-	○	○	○	○
L system	○	○	-	-	○	○	○	○

The following servo data and spindle data can be displayed as waveforms. Data can be displayed for two channels simultaneously in 1-hour increments on a continuous basis or on a one-shot basis.

Servo data	Spindle data
Current feedback	Motor load
Current command	Motor rotation speed (speed command value)
Position deviation	Position deviation
Position command	Position command
Speed feedback	Speed feedback (r/min)
	Synchronous tap error width (μm)
	Synchronous tap error angle (0.001 degree)

15.4.5 Machine Operation History Monitor

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

Up to 256 past key inputs on the operation board and changes in the input signals are recorded. The history contents can be viewed on the history screen, and the data is retained even after the power has been turned OFF.

15 Safety and Maintenance

15.4 Maintenance and Troubleshooting

15.4.6 NC Data Backup

This function serves to back up the parameters and other data of the NC control unit.
The data can also be restored.

(1) RS-232C

Model \ M/L system	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

[Backup target]

Machining programs, parameters, workpiece offset data, common variables, tool compensation data, tool life control data
Ladders (ladder, message)
SRAM data

(2) Ethernet

Model \ M/L system	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	△	△	△	△
L system	-	-	-	-	△	△	△	△

[Backup target]

Machining programs, parameters, common variables, tool compensation data, tool life control data
Ladders (ladder, message)
SRAM data

(3) Cassette memory

Model \ M/L system	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	○	○	○	○
L system	-	-	-	-	○	○	○	○

The memory cassette for maintenance is used to back up and restore the NC data.

Model	Compatible memory cassette for maintenance
M64AS	HR410/HR450
M64S	HR411/HR451
M65S/M66S	HR415/HR455

[Backup targets ... The following data is backed up in a batch.]

Ladders (ladder, message)
SRAM data

(4) IC card

Model \ M/L system	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	△	△	△	△
L system	-	-	-	-	△	△	△	△

[Backup target]

Machining programs, parameters, common variables, tool compensation data, tool life control data
Ladders (ladder, message)
SRAM data

15 Safety and Maintenance

15.4 Maintenance and Troubleshooting

15.4.7 PLC I/F Diagnosis

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

When the **I/F DIAGN** menu key is pressed, the PLC interface diagnosis screen appears. The input and output signals for PLC control can be displayed and set on this screen. This function can be used to check the machine sequence operations for PLC development, check the input/output data between the control unit and PLC when trouble occurs in operation, initiate forced definitions, and so on.

15 Safety and Maintenance

15.4 Maintenance and Troubleshooting

15.4.8 MELDASNET

15.4.8.1 Machine Builder Network System

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	Δ	Δ	Δ	Δ
L system	-	-	-	-	Δ	Δ	Δ	Δ

With the machine builder network system, using the software for personal computer "MELDAS remote diagnosis tool kit" which Mitsubishi Electric Corporation sells separately to machine builders, NC is connected with the personal computer of the machine builder by the communication circuit, and the diagnosis data file required for the service is transmitted and received.

15.4.8.2 Anshin-net

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	Δ	Δ	Δ	Δ
L system	-	-	-	-	Δ	Δ	Δ	Δ

By means of a network that connects the CNC systems of end users all over Japan with the Mitsubishi Electric Machine Tool Engineering (RKE) Call Center, detailed information services and speedier maintenance support can be provided.

A separate agreement must be signed with RKE in order to receive this service.

(Note) This service is applied only in Japan.

Service menu

(1) Peace-of-mind menu		
Automatic notification of alarms	Standard	The Call Center is aware of the circumstances under which alarms have occurred during automatic operation. Speedy support can be provided even during unmanned operation.
Single-action call	Standard	Simply by pressing the single-action button, the machine status information can be transmitted so that technical support can be received from expert engineers.
Periodical monitoring	Standard	Malfunctioning is prevented by monitoring and analyzing the warning messages on a periodical basis.
Backup	Standard	Speedy action is taken to deal with even the loss of data in the memory by backing up the parameter data on a periodical basis.
(2) Handy menu		
Operator notification	Standard	The operator is automatically notified of the completion of machining, error occurrence such as operation stop, or other trouble by voice messages to the operator's cell phone, etc.
Periodical monitoring report	Standard	Transmission of written reports compiled by totalizing the monitored data comes in handy for managing and formulating production plans.
Remote monitoring	Optional	The operating conditions of the machine can be ascertained at any time by simply accessing the Call Center from the user's telephone.
Machining data sharing	Optional	The Call Center's servers are released to users who can then utilize them as media for storing programs.

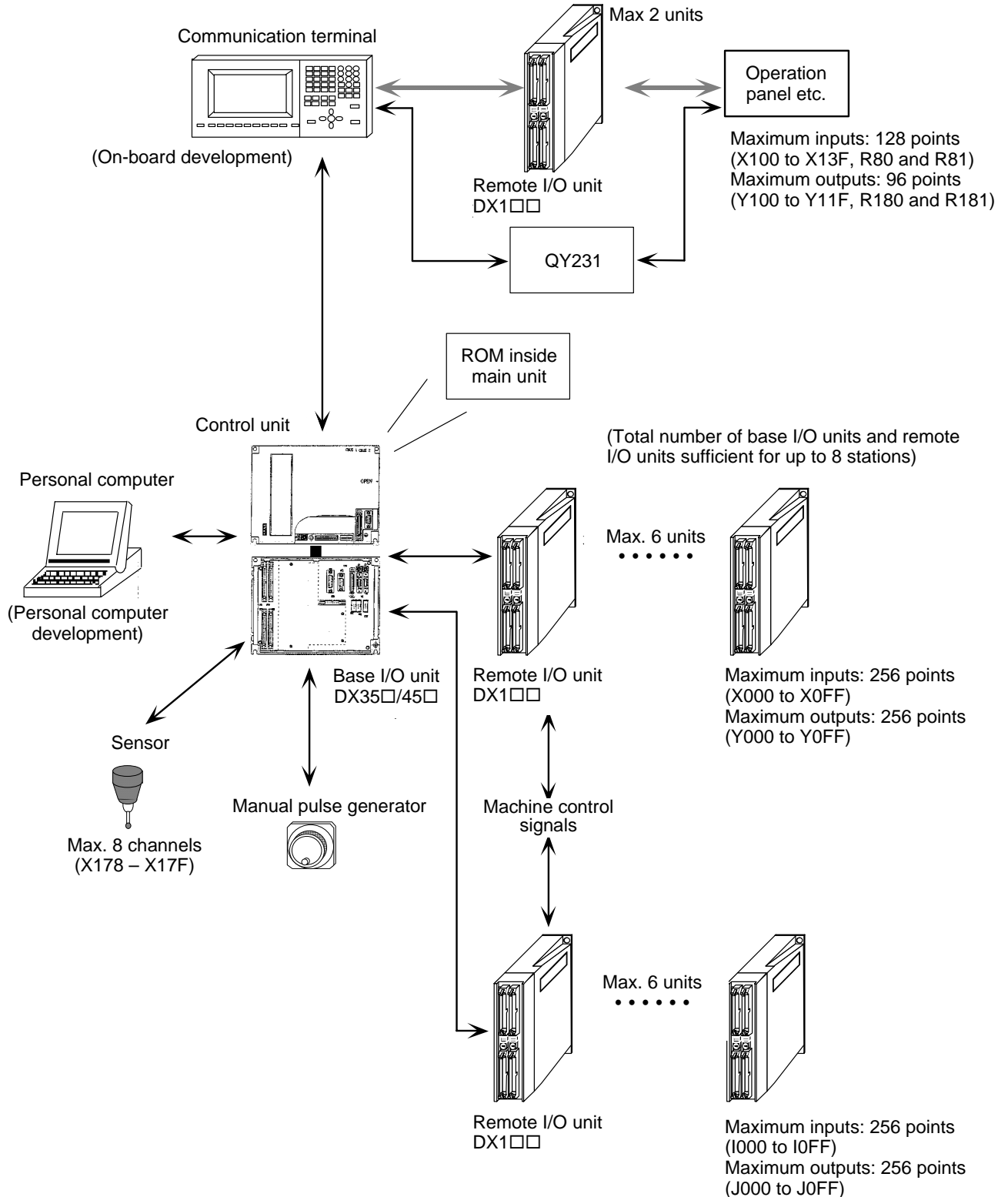
16 Cabinet and Installation

16.1 Cabinet Construction

16 Cabinet and Installation

16.1 Cabinet Construction

The configuration of the unit used by the MELDAS 60/60S series is shown below.
Refer to the Connection / Maintenance Manual for details.



16 Cabinet and Installation

16.1 Cabinet Construction

List of configuration units

(1) Control unit

Type	Function	Configuration element	Details
FCU6-MU011	M64 control unit set FCA64A-B, FCA64-B system compatible unit	Main control card (HR113) 24V input power supply card (HR083) Communication card (HR531) Case set	Export Trade Control Ordinance and Foreign Trade Ordinance noncompliant unit
FCU6-MU021	M65 control unit set FCA65-A, FCA65-P1 system compatible unit	Main control card (HR114) 24V input power supply card (HR083) Communication card (HR171) Case set	Export Trade Control Ordinance and Foreign Trade Ordinance noncompliant unit
FCU6-MU032	M64AS/64S/65S control unit set FCA64AS/64S/65S system compatible unit	Main control card (HR116) 24V input power supply card (HR083) Communication card (HR171) Case set	Export Trade Control Ordinance and Foreign Trade Ordinance noncompliant unit M64AS is used as a set with FCU6-HR410 M64S is used as a set with FCU6-HR411 M65S is used as a set with FCU6-HR415
FCU6-MA031	M66/M66S control unit set FCA66-A/FCA66S system compatible unit	Main control card (HR146) 24V input power supply card (HR083) Communication card (HR171) Case set	Export Trade Control Ordinance and Foreign Trade Ordinance compliant unit M66S is used as a set with FCU6-HR415

(2) Control unit options

Type	Function	Configuration element	Details
HR513	External PLC link II (bus connection, interface) Mounted into control unit's extension slot	Control card (HR513)	Dedicated for MELDAS60/60S series Type connected to MELSEC A1S, A N, A A, A U Series
HR571	External PLC link I (M-NET interface) Mounted into control unit's extension slot	Control card (HR571)	Dedicated for MELDAS60 series Type connected to MELSEC AJ71C22 unit (serial link)
FCU6-EP203-1	(1) High-speed program server (2) Data input/output { Ethernet communication } IC card	Control card for IC card (HR831) Communication card for Ethernet (HR832) Connection cable	(1) Dedicated for MELDAS64S/ 65/65S/66/66S (M system) (2) Dedicated for MELDAS60S series ATA memory card interface + Ethernet interface
HR576	CC-Link card	Control card (HR576)	Dedicated for MELDAS60S series

(3) Communication terminal

Type	Function	Configuration element	Details
FCU6-DUT32	10.4-type monochrome LCD display unit (separated type)	10.4-type LCD, Escutcheon Control card (RX215)	Used as a set with FCUA-KB20. Control card 24VDC input
FCU6-DUN33	10.4-type color LCD display unit (separated type)	10.4-type LCD, Escutcheon Control card (RX215)	Used as a set with FCUA-KB20. Control card 24VDC input
FCUA-LD100	7.2-type monochrome LCD liquid crystal (integrated type)	7.2-type LCD, Escutcheon Control card (RX213) Key switches	Control card 24VDC input
FCUA-LD10	7.2-type monochrome LCD display unit (separated type)	7.2-type LCD, Escutcheon Control card (RX213)	Used as a set with FCUA-KB20. Control card 24VDC input
FCUA-CT100	9-type monochrome CRT + communication terminal (integrated type, machining center system sheet)	9-type monochrome CRT, Escutcheon Control card (RX211) Key switches	Control card 24VDC input CRT section 100VAC input
FCUA-CT120	9-type monochrome CRT + communication terminal (integrated type, lathe system sheet)	9-type monochrome CRT, Escutcheon Control card (RX211) Key switches	Control card 24VDC input CRT section 100VAC input

16 Cabinet and Installation

16.1 Cabinet Construction

Type	Function	Configuration element	Details
FCUA-CR10	9-type monochrome CRT communication terminal (separated type)	9-type monochrome CRT, Escutcheon	Used as a set with FCUA-KB10/KB12. Control card 24VDC input CRT section 100VAC input
FCUA-KB10	Communication terminal (separated type, machining center system sheet)	Key switch Control card	Used as a set with FCUA-CR10.
FCUA-KB12	Communication terminal (separated type, lathe system sheet)	Key switch Control card	Used as a set with FCUA-CR10.
FCUA-KB20	Communication terminal (separated type, machining center system sheet)	Key switch	Use as a set with FCU6-DUT32/DUN33 or FCUA-LD10
FCUA-KB021	Communication terminal (separated type, machining center system sheet)	Key switch	Use as a set with FCU6-DUT32/DUN33 KB20 with changed outline dimensions
FCUA-KB30	Communication terminal (separated type, lathe system sheet)	Key switch	Use as a set with FCU6-DUT32/DUN33 or FCUA-LD10
FCUA-KB031	Communication terminal (separated type, lathe system sheet)	Key switch	Use as a set with FCU6-DUT32/DUN33 KB30 with changed outline dimensions

(4) Base I/O unit

Type	Function	Configuration element	Details
FCU6-DX350	Sink input + sink output base I/O unit	I/O card (HR325) Aluminum panel for panel installation Additional I/O card (HR211)	DI/DO input/output sink input 48 points + sink output 48 points Synchronous feed encoder interface 1ch, Skip input 8 points Remote I/O unit interface 2ch Servo drive unit interface 2 part systems RS-232C device 1ch, manual pulse generator 3ch
FCU6-DX351	Source input + source output base I/O unit	I/O card (HR335) Aluminum panel for panel installation Additional I/O card (HR211)	DI/DO input/output source input 48 points + source output 48 points Synchronous feed encoder interface 1ch, Skip input 8 points Remote I/O unit interface 2ch Servo drive unit interface 2 part systems RS-232C device 1ch, manual pulse generator 3ch
FCU6-DX450	Sink input + sink output base I/O unit	I/O card (HR327) Aluminum panel for panel installation Additional I/O card (HR211)	DI/DO input/output sink input 64 points + sink output 64 points Synchronous feed encoder interface 1ch, Skip input 8 points Remote I/O unit interface 2ch Servo drive unit interface 2 part systems RS-232C device 1ch, manual pulse generator 3ch
FCU6-DX451	Source input + source output base I/O unit	I/O card (HR337) Aluminum panel for panel installation Additional I/O card (HR211)	DI/DO input/output source input 64 points + source output 64 points Synchronous feed encoder interface 1ch, Skip input 8 points Remote I/O unit interface 2ch Servo drive unit interface 2 part systems RS-232C device 1ch, manual pulse generator 3ch
FCU6-HR377	Source input + 200mA source output base I/O unit	I/O card (HR377) Reinforcing fitting	DI/DO input/output source input 64 points + source output 64 points Synchronous feed encoder interface 1ch, Skip input 8 points Remote I/O unit interface 2ch Servo drive unit interface 2 part systems RS-232C device 1ch, manual pulse generator 3ch
FCU6-HR378	Source input + 200mA source output base I/O unit Common separated, with output protection fuse	I/O card (HR378) Reinforcing fitting	DI/DO input/output source input 64 points + source output 64 points Synchronous feed encoder interface 1ch, Skip input 8 points Remote I/O unit interface 2ch Servo drive unit interface 2 part systems RS-232C device 1ch, manual pulse generator 3ch

16 Cabinet and Installation

16.1 Cabinet Construction

(5) Remote I/O unit

Type	Function	Configuration element	Details
FCUA-DX100	Sink/source input + sink output	RX311	DI/DO = 32 points/32 points
FCUA-DX110	Sink/source input + sink output	RX311 + RX321-1	DI/DO = 64 points/48 points
FCUA-DX120	Sink/source input + sink output + analog output	RX311 + RX321	DI/DO = 64 points/48 points + analog output 1 point
FCUA-DX140	Sink/source input + sink output + analog input/output	RX311 + RX341	DI/DO = 32 points/32 points + analog input 4 points + analog output 1 point
FCUA-DX101	Sink/source input + source output	RX312	DI/DO = 32 points/32 points
FCUA-DX111	Sink/source input + source output	RX312 + RX322-1	DI/DO = 64 points/48 points
FCUA-DX121	Sink/source input + source output + analog output	RX312 + RX322	DI/DO = 64 points/48 points + analog output 1 point
FCUA-DX141	Sink/source input + source output + analog input/output	RX312 + RX341	DI/DO = 32 points/32 points + analog input 4 points + analog output 1 point

(6) Scan I/O card

Type	Function	Configuration element	Details
HR357	Scan I/O (source)	HR357	Scan DI/DO = 64 points/64 points DI/DO = 32 points /32 points
HR347	Scan I/O (sink)	HR347	Scan DI/DO = 64 points /64 points DI/DO = 32 points /32 points

(7) Extended I/O card

Type	Function	Configuration element	Details
QY231	Sink/source input + source output	QY231	Sink/source input 64 points + source output 48 points

16 Cabinet and Installation

16.2 Power Supply, Environment and Installation Conditions

16.2 Power Supply, Environment and Installation Conditions

⚠ Caution

- ⚠ Follow the power supply specifications (input voltage range, frequency range, momentary power failure time range) described in this manual.
- ⚠ Follow the environment conditions (ambient temperature, humidity, vibration, ambient atmosphere) described in this manual.

(1) Environment conditions in control part

Unit name		Control unit	
Type		FCU6-MU011/MU021/MU032/MA031	
General specifications	Ambient temperature	During operation	0 to 55°C
		During storage	-20 to 60°C
	Ambient humidity	During operation	Long term, to 75% RH (with no dew condensation) Short term, to 95% RH (with no dew condensation) (Note 1)
		During storage	to 75% RH (with no dew condensation)
	Vibration resistance		4.9m/s ² or less (during operation)
	Shock resistance		29.4m/s ² or less (during operation)
Working atmosphere		No corrosive gases, dust or oil mist	
Power specifications	Power voltage		24VDC±5% Ripple ±5% (P-P)
	Instantaneous stop tolerance time		Depends on the specifications of the 24VDC power supply used.
	Current consumption		1.5A
Heating value		20W (Standard specification)	
Mass		1.1kg	
Unit size		Refer to Appendix.	

(Note 1) The period is within one month.

(2) Environment conditions in electric cabinet

Unit name		Base I/O unit		
Type		FCU6-DX350/351/450/451	FCU6-HR377/378	
General specifications	Ambient temperature	During operation	0 to 55°C	
		During storage	-20 to 60°C	
	Ambient humidity	During operation	Long term, to 75% RH (with no dew condensation) Short term, to 95% RH (with no dew condensation) (Note 1)	
		During storage	to 75% RH (with no dew condensation)	
	Vibration resistance		4.9m/s ² or less (during operation)	
	Shock resistance		29.4m/s ² or less (during operation)	
Working atmosphere		No corrosive gases, dust or oil mist		
Power specifications	Power voltage	24VDC±5% Ripple ±5% (P-P)	5VDC±5%	
	Current consumption	24V 1.2A (Note 2) 24V max. 5.0A (Note 3)	24V 13A (Note 3) 5V 1.0A (when using as second unit)	
Heating value		Max. 30W (Note 3)	Max. 50W (Note 3)	
Mass		2kg		
Unit size		Refer to Appendix.	195mm (W) × 280mm (H)	

(Note 1) The period is within one month.

(Note 2) Amount consumed by control circuit

(Note 3) Differs according to the number of machine input operation points and the load and number of points connected to the machine output. The maximum value applies when all points are ON.

16 Cabinet and Installation

16.2 Power Supply, Environment and Installation Conditions

(3) Communication terminal

Unit name		Communication terminal						
Type		FCUA-CT 100/120	FCUA- CR10	FCUA- KB10/KB12	FCUA-KB20/30 FCU6-KB021/031	FCUA- LD10/ 100	FCU6-DUT32 FCU6-DUN33	
General specifications	Ambient temperature	During operation	0 to 55°C				0 to 50°C	
		During storage	-20 to 65°C				-20 to 60°C	
	Ambient humidity	During operation	Long term, to 75% RH (with no dew condensation) Short term, to 95% RH (with no dew condensation) (Note 1)					
		During storage	to 75% RH (with no dew condensation)					
	Vibration resistance		4.9m/s ² or less (during operation)					
	Shock resistance		29.4m/s ² or less (during operation)					
	Working atmosphere		No corrosive gases, dust					
Power noise		1kV (P-P)						
Power specifications	Power voltage		Single-phase 100 VAC to 115 VAC -15% +10% 50/60Hz ±5% Ripple ±5% (P-P)		24VDC±5% Ripple ±5% (P-P)			
			24VDC ±5%	—				
	Instantaneous stop tolerance time		20ms or less					
Current consumption		100V, 0.4A 24V, 0.6A	100V, 0.4A	24V, 0.6A	—	24V, 0.9A	24V, 0.9A	
Heating value		55W	40W	15W	—	20W	20W	
Mass		4.8kg	4.2kg	0.7kg	0.4kg	1.2kg	1.8kg	
Unit size		Refer to Appendix.						

(Note 1) The period is within one month.

(4) Remote I/O unit

Unit name		Remote I/O unit				
Type		FCUA-DX10□	FCUA-DX11□	FCUA-DX12□	FCUA-DX14□	
General specifications	Ambient temperature	During operation	0 to 55°C			
		During storage	-20 to 65°C			
	Ambient humidity	During operation	Long term, to 75% RH (with no dew condensation) Short term, to 95% RH (with no dew condensation) (Note 1)			
		During storage	to 75% RH (with no dew condensation)			
	Vibration resistance		4.9m/s ² or less (during operation)			
	Shock resistance		29.4m/s ² or less (during operation)			
	Working atmosphere		No corrosive gases, dust			
Power noise		1kV (P-P)				
Power specifications	Power voltage		24VDC±5% Ripple ±5% (P-P)			
	Instantaneous stop tolerance time		—			
	Current consumption		24V 0.7A (Note 2)	24V 1.5A (Note 2)	24V 0.7A (Note 2)	
Heating value		Max. 25W (Note 3)	Max. 30W (Note 3)	Max. 30W (Note 3)		
Mass		470g	570g	590g	550g	
Unit size		Refer to Appendix.				

(Note 1) The period is within one month.

(Note 2) Amount consumed by control circuit

(Note 3) Differs according to the number of machine input operation points and the load and number of points connected to the machine output. The maximum value applies when all points are ON.

16 Cabinet and Installation

16.2 Power Supply, Environment and Installation Conditions

(5) Servo / Spindle

Refer to the following manuals for details on the servo and spindle system.

MDS-C1 Series	Specification Manual	(BNP-C3040)
MDS-CH Series	Specification Manual	(BNP-C3044)
MDS-B-SVJ2 Series	Specifications and Instruction Manual	(BNP-B3937)
MDS-B Series		
Linear Servo System	Specifications and Instruction Manual	(BNP-B3977)
MDS-B-SPJ2 Series	Specification Manual	(BNP-B2164)
MDS-J2-CT Series	Specifications and Instruction Manual	(BNP-B3944)

17 Servo/Spindle System

17.1 Feed Axis

17 Servo/Spindle System

Refer to the following manuals for details on the servo and spindle system.

MDS-C1 Series	Specification Manual	(BNP-C3040)
MDS-CH Series	Specification Manual	(BNP-C3044)
MDS-B-SVJ2 Series	Specifications and Instruction Manual	(BNP-B3937)
MDS-B Series		
Linear Servo System	Specifications and Instruction Manual	(BNP-B3977)
MDS-B-SPJ2 Series	Specification Manual	(BNP-B2164)
MDS-J2-CT Series	Specifications and Instruction Manual	(BNP-B3944)

17.1 Feed Axis

17.1.1 MDS-C1-V1/C1-V2 (200V)

(1) Servo motor: HC□□-A51/E51 (1000kp/rev)

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	Δ	Δ	Δ	Δ
L system	-	-	-	-	Δ	Δ	Δ	Δ

(2) Servo motor: HC□□-A42/E42 (100kp/rev)

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	□	□	□	□	□	□	□	□
L system	□	□	□	□	□	□	□	□

17.1.2 MDS-B-V14L (200V)

(1) Linear servo motor: LM-NP

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	□	□	□	□	□	□	□	□
L system	□	□	□	□	□	□	□	□

17.1.3 MDS-CH-V1/CH-V2 (400V)

(1) Servo motor: HC□□-A51/E51 (1000kp/rev)

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	Δ	Δ	Δ	Δ
L system	-	-	-	-	Δ	Δ	Δ	Δ

(2) Servo motor: HC□□-A42/E42 (100kp/rev)

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	□	□	□	□	Δ	Δ	Δ	Δ
L system	□	□	□	□	Δ	Δ	Δ	Δ

(3) Linear servo motor: LM-NP

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	□	□	□	□	□	□	□	□
L system	□	□	□	□	□	□	□	□

17 Servo/Spindle System

17.1 Feed Axis

17.1.4 MDS-B-SVJ2 (Compact and Small Capacity)

(1) Servo motor: HC□□-A42/E42 (100kp/rev)

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
M system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
L system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

(2) Servo motor: HC□□-A47 (100kp/rev)

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
M system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
L system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

(3) Servo motor: HC□□-A33/E33 (25kp/rev)

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
M system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
L system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

17 Servo/Spindle System

17.2 Spindle

17.2 Spindle

17.2.1 MDS-C1-SP/C1-SPM/B-SP (200V)

(1) Spindle motor: SJ/SJ-V

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
M system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
L system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

(2) IPM Spindle motor: SJ-PMF

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
M system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
L system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

17.2.2 MDS-CH-SP/CH-SPH (400V)

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
M system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
L system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

17.2.3 MDS-B-SPJ2 (Compact and Small Capacity)

(1) Spindle motor: SJ-P/SJ-PF

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
M system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
L system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

17.3 Auxiliary Axis

17.3.1 Index/Positioning Servo: MR-J2-CT

(1) Servomotor: HC-SF/HC-RF (16kp/rev)

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
M system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
L system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

(2) Servomotor: HA-FF/HC-MF (8kp/rev)

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
M system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
L system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

17 Servo/Spindle System

17.4 Power Supply

17.4 Power Supply

17.4.1 Power Supply: MDS-C1-CV/B-CVE

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
M system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
L system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

17.4.2 AC Reactor for Power Supply

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
M system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
L system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

17.4.3 Ground Plate

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
M system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
L system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

17.4.4 Power Supply: MDS-A-CR (Resistance Regeneration)

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
M system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
L system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

18 Machine Support Functions

18.1 PLC

18 Machine Support Functions

18.1 PLC

18.1.1 PLC Basic Function

18.1.1.1 Built-in PLC Basic Function

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

(1) Ladder commands

Basic commands (bit processing commands)

20 commands including LD, LDI, OR, ORI, AND, ANI, OUT, PLS, etc.

Function commands

76 commands including data transfer, 4 basic arithmetic operations, logic arithmetic operations, large/small identification, binary/BCD conversion, branching, conditional branching, decoding, encoding, etc.

Exclusive commands

ATC control commands, and 14 others

Tool life management

(2) Devices

The table below lists the devices which can be used by the PLC. (PLC4B)

Device	Device range	Unit	Details	Remarks
X	X0 to X4BF (1216 points)	1 bit	Input signal to PLC Machine input, etc.	
Y	Y0 to Y53F (1344 points)	1 bit	Output signal from PLC Machine output, etc.	
U	U0 to U17F (384 points)	1 bit	Input signal to PLC Signal used in the second part system	
W	W0 to W1FF (512 points)	1 bit	Output signal from PLC Signal used in the second part system	
I	I0 to I3FF (1024 points)	1 bit	Input signal to PLC	
J	J0 to J63F (1600 points)	1 bit	Output signal from PLC	
S	S0 to S1F (32 points)	1 bit	Input signal to PLC	
	S40 to S5F (32 points)	1 bit		
	S80 to S9F (32 points)	1 bit		
	SC0 to SDF (32 points)	1 bit		
	S100 to S13F (64 points)	1 bit		
S	S20 to S3F (32 points)	1 bit	Output signal from PLC	
	S60 to S7F (32 points)	1 bit		
	SA0 to SBF (32 points)	1 bit		
	SE0 to SFF (32 points)	1 bit		
M	M0 to M5119 (5120 points)	1 bit	Temporary memory	
G	G0 to G3071 (3072 points)	1 bit	Temporary memory	
F	F0 to F127 (128 points)	1 bit	Temporary memory, Alarm message interface	
L	L0 to L255 (256 points)	1 bit	Latch relay (backup memory)	
E	E0 to E127 (128 points)	1 bit	Special relay	

18 Machine Support Functions

18.1 PLC

Device	Device range	Unit	Details	Remarks
T	T0 to T15 (16 points)	1 bit/16 bits	10 ms unit timer	
	T16 to T95 (80 points)	1 bit/16 bits	100 ms unit timer	
	T96 to T103 (8 points)	1 bit/16 bits	100 ms integral timer	
Q	Q0 to Q39 (40 points)	1 bit/16 bits	10 ms unit timer (fixed timer)	
	Q40 to Q135 (96 points)	1 bit/16 bits	100 ms unit timer (fixed timer)	
	Q136 to Q151 (16 points)	1 bit/16 bits	100 ms integral timer (fixed timer)	
C	C0 to C23 (24 points)	1 bit/16 bits	Counter	
B	B0 to B103 (104 points)	1 bit/16 bits	Counter (fixed counter)	
D	D0 to D1023 (1024 points)	16 bits/ 32 bits	Data register Register for arithmetic operations	
R	R0 to R8191 (8192 points)	16 bits/ 32 bits	File register For interface between PLC and control unit. 10ms timer expansion (400 points)	
A	A0, A1 (2 points)	16 bits/ 32 bits	Accumulator	
Z	— (1 point)	16 bits	For D or R address indexing (for ±n)	
V	— (1 point)	16 bits	For D or R address indexing (for ±n)	
N	N0 to N7 (8 points)	—	Master controller nesting level	
P	P0 to P255 (256 points)	—	Label for conditional jump and subroutine call commands	
K	K-32768 to K32767	—	Decimal constant for 16-bit command	
	K-2147483648 to K2147483647	—	Decimal constant for 32-bit command	
H	H0 to HFFFF	—	Hexadecimal constant for 16-bit command	
	H0 to HFFFFFFFF	—	Hexadecimal constant for 32-bit command	

18 Machine Support Functions

18.1 PLC

Device No. when using PLC4B	Device No. when using GX Developer
X0 to X4BF	X0 to X4BF
U0 to U17F	X4C0 to X63F
I0 to I3FF	X640 to XA3F
S0 to S1F S40 to S5F S80 to S9F SC0 to SDF S100 to S13F	XA40 to XAFF
Y0 to Y53F	Y0 to Y53F
W0 to W1FF	Y540 to Y73F
J0 to J63F	Y740 to YD7F
S20 to S3F S60 to S7F SA0 to SBF SE0 to SFF	YD80 to YDFF
M0 to M5119	M0 to M5119
G0 to G3071	M5120 to M8191
F0 to F127	F0 to F127
L0 to L255	L0 to L255
E0 to E127	SM0 to SM127
T0 to T15	T0 to T15
Q0 to Q39	T16 to T55
T16 to T95	T56 to T135
Q40 to Q135	T136 to T231
T96 to T103	T232 to T239
Q136 to Q151	T240 to T255
C0 to C23	C0 to C23
B0 to B103	C24 to C127
D0 to D1023	D0 to D1023
R0 to R8191	R0 to R8191
A0, A1	
Z	Z0
V	Z1
N0 to N7	N0 to N7
P0 to P255	P0 to P255
K-32768 to K32767	K-32768 to K32767
K-2147483648 to K2147483647	K-2147483648 to K2147483647
H0 to HFFFF	H0 to HFFFF
H0 to HFFFFFFF	H0 to HFFFFFFF

18 Machine Support Functions

18.1 PLC

(3) External alarm messages

The contents of the alarms which have occurred during sequence (user PLC) processing can be displayed on the setting and display unit.

Up to four alarm message displays can be displayed simultaneously on the alarm diagnosis screen. The maximum length of one message is 32 characters.

(4) External operator messages

When a condition has arisen in which a message is to be relayed to the operator, an operator message can be displayed separately from the alarm message.

The maximum length of an operator message on the alarm diagnosis screen is 60 characters. The number of messages displayed at the same time is one.

(5) PLC switches

32 points of PLC switches can be set on the setting and display unit screen, and the ON/OFF control executed. The switches can be used as part of the machine operation switches. The switch applications can be freely determined with the sequence program, and each switch name can be created with the PLC and displayed on the setting and display unit.

(6) Load meter display

A load meter can be displayed on the setting and display unit.

Up to two axes designated with the built-in PLC such as the spindle load and Z axis load can be displayed as bar graphs on the screen.

(7) Timer / counter setting display

(a) PLC timer

The setting value of the timer used by the built-in PLC can be set from the screen on the setting and display unit.

The timer types include the 10ms, 100ms and 100ms integral types.

Whether to validate the timer in the PLC program or to validate the setting value from the screen can be selected with the parameters.

Whether to hold the integral timer when the power is turned OFF can also be selected.

(b) PLC counter

The setting value of the counter used by the built-in PLC can be set from this screen.

Whether to validate the constants in the PLC program or to validate the setting value from the screen can be selected with the parameters.

Whether to hold the counter value when the power is turned OFF can also be selected.

(8) PLC parameter setting display

The PLC constants set with the data type and the bit selection parameters set with bit types can be set from the screen as parameters used by the built-in PLC.

(a) PLC constants

There are PLC constants that can be set with data types as parameters used by the built-in PLC. The set data is set in the R register of the PLC and backed up. If data is set in the R register corresponding to the PLC constant with sequence program MOV commands, etc., the data will be backed up. However, the display will not change, so enter another screen, and then select this screen again.

Up to 48 items can be set, and the setting range is ± 8 digits.

(b) Bit selection parameters

There are bit selection parameters set with bit types as parameters used by the built-in PLC. The set data is set in the R register of the PLC and backed up.

When using bit operation in the sequence program, the details of the R register are transferred to the temporary memory (M) with the MOV command. If the data is set in the R register corresponding to the bit selection with the MOV command, etc., the data will be backed up. However, the display will not change, so enter another screen and then select this screen again.

18 Machine Support Functions

18.1 PLC

(9) External key input

By inputting the key data from the built-in PLC, the same operation as when the operator operates the operation board can be done.

(10) Real spindle speed output

The real spindle speed is converted by the signals of the encoder installed on the spindle and is output to the PLC. The output increment is 0.001r/min.

(11) Workpiece counter display

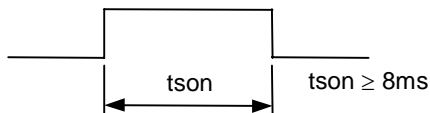
The number of workpieces can be set and displayed when continuously machining workpieces. The M code to be count, the current number of machined workpieces and the max. machining value is set with parameters.

This data can be read by the user PLC (when built-in PLC specifications are used), and the number of machined workpieces can be controlled. A signal will be output to the PLC when the counted number reaches the set max. value.

(12) High speed input/output signal

There are signals that can be input and output at a 7.1ms cycle for high-speed processing.

(a) Input signal ON time



(b) After the signal output is set in the interface, it can be output to the machine side with a max. 7.1ms delay. The input also appears on the interface with a 7.1ms delay.

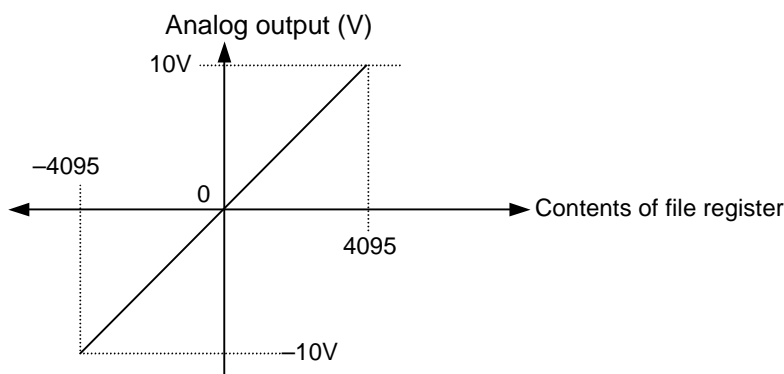
(c) The signals used for high-speed processing are assigned with the parameters. Assignment is possible in a continuous 16-point unit.

(13) PLC analog voltage control

(a) Analog output

When the specified data is put in the file register, the corresponding analog voltage is output from the analog output external connector.

<Relationship between file register contents and analog output voltage>



Output voltage	0 to $\pm 10V$ ($\pm 5\%$)
Resolution	Full scale (10V)/4095
Load condition	10 k Ω resistance load (standard)
Output impedance	220 Ω

(Note) The remote I/O unit DX120/DX121 is required for analog output.

18 Machine Support Functions

18.1 PLC

18.1.2 Built-in PLC Processing Mode

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

An exclusive sequence program that controls the various signals between the controller and machine to realize operation applicable to each machine must be created.

The sequence execution modes include high-speed processing and main processing.

(1) High-speed processing

This mode provides repeated execution at 7.1ms cycles. It is used to process signals requiring high speeds.

The max. number of program steps for high-speed processing (1 period) is 150 steps when using basic commands.

(2) Main processing

This mode provides normal sequence processing. The processing cycle depends on the number of sequence steps.

18.1.2.1 PLC4B I/F

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

This function enables PLC development using the conventional PLC development tool (PLC4B) for MELDAS systems

18.1.2.2 MELSEC Development Tool I/F

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	-	-	○	○	○	○
L system	○	○	-	-	○	○	○	○

This function enables the data of the PLC contained inside the NC system to be developed and debugged using the GX Developer installed in a personal computer (OS:Windows).

Many and varied functions of the GX Developer make it possible to reduce the PLC data development and debugging time.

18 Machine Support Functions

18.1 PLC

18.1.3 Built-in PLC Capacity (Number of Steps)

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	16000	32000	32000	32000	32000	32000	32000	32000
L system	16000	32000	32000	32000	32000	32000	32000	32000

18.1.4 Machine Contact Input/Output I/F

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

Caution

- ① Follow the remote type machine contact input/output interface described in this manual.
(Connect a diode in parallel with the inductive load or connect a protective resistor in serial with the capacitive load, etc.)

Refer to the MELDAS 60/60S Series Connection/Maintenance Manual for details.

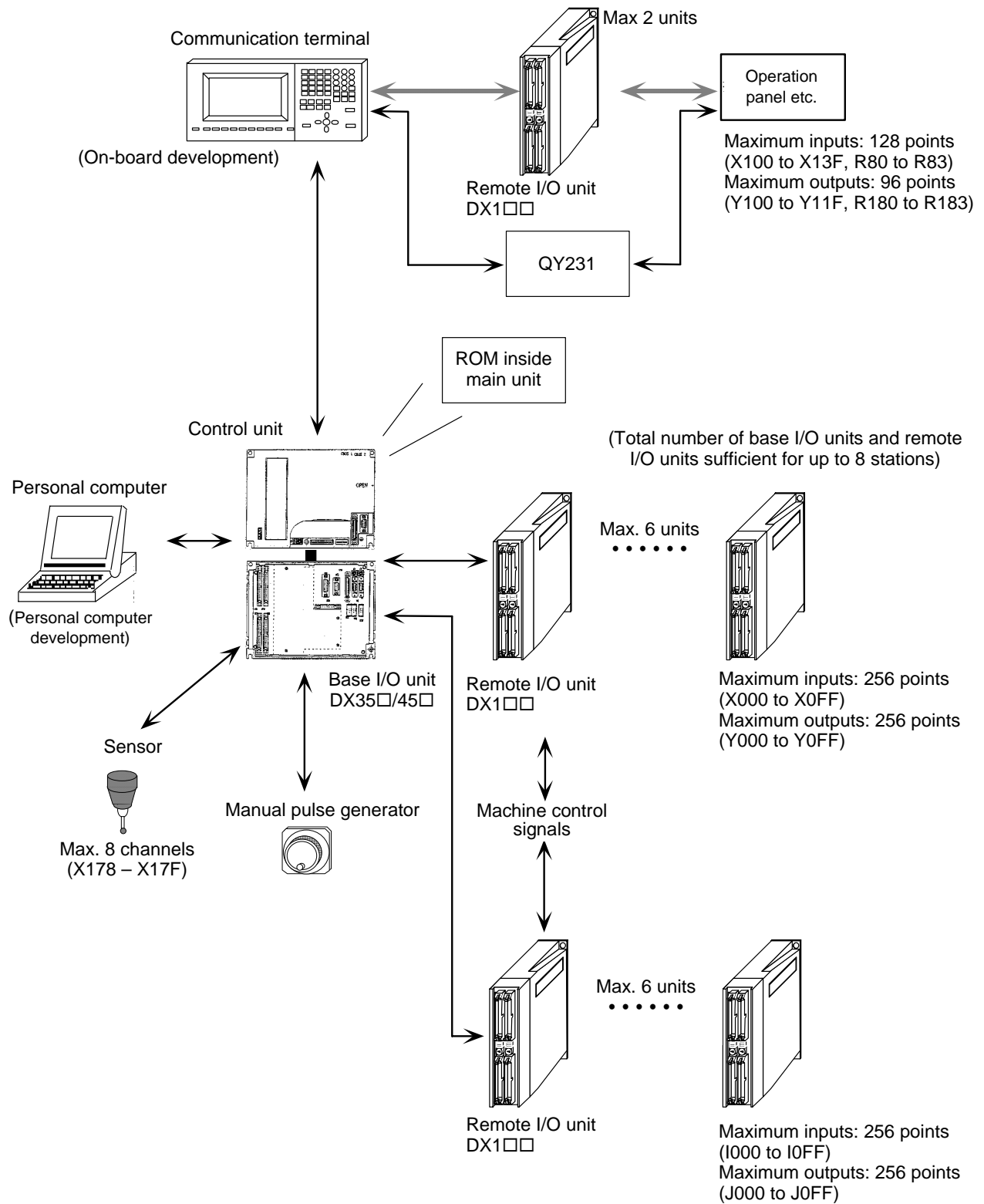
(1) Types of remote I/O units

The remote I/O units (FCUA-DX□□□) are shown in the remote I/O unit list according to the types of signals that can be input/output and the number of contacts. There are 8 types, and are serially linked (MC link B) to the control unit or communication terminal when used.

Multiple remote I/O units can be combined for use if the total of possessed channel during the serial link connection is less than eight.

18 Machine Support Functions

18.1 PLC



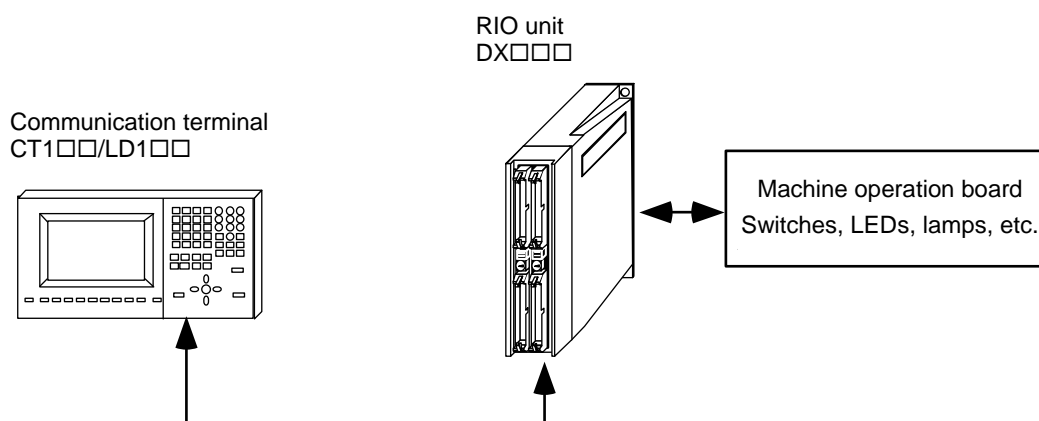
18 Machine Support Functions

18.1 PLC

Remote I/O unit list

Type	Function	Configuration element	Details
FCUA-DX100	Sink/source input + sink output	RX311	DI/DO = 32 points/32 points
FCUA-DX110	Sink/source input + sink output	RX311 + RX321-1	DI/DO = 64 points/48 points
FCUA-DX120	Sink/source input + sink output + analog output	RX311 + RX321	DI/DO = 64 points/48 points + analog output 1 point
FCUA-DX140	Sink/source input + sink output + analog input/output	RX311 + RX341	DI/DO = 32 points/32 points + analog input 4 points + analog output 1 point
FCUA-DX101	Sink/source input + source output	RX312	DI/DO = 32 points/32 points
FCUA-DX111	Sink/source input + source output	RX312 + RX322-1	DI/DO = 64 points/48 points
FCUA-DX121	Sink/source input + source output + analog output	RX312 + RX322	DI/DO = 64 points/48 points + analog output 1 point
FCUA-DX141	Sink/source input + source output + analog input/output	RX312 + RX341	DI/DO = 32 points/32 points + analog input 4 points + analog output 1 point

Operation board remote I/O unit



The remote I/O unit (hereafter RIO unit) connected to the main unit can also be connected to the remote terminal.

As when it is connected to the main unit, the numbers are allocated to the devices using the rotary switches.

The switches are set separately and the settings are not related to the RIO unit which is used by the main unit.

Rotary switch number	Device numbers which are loaded	Device numbers which are output	Analog output (AO)
0	X100 to X11F	Y100 to Y11F (Y10F)	None
1	X120 to X13F	Y120 to Y13F (Y12F)	
2	R80, R81	R180, R181	
3	R82, R83	R182, R183	

The figures in parentheses denote the range of the devices when they have been installed on the right side of the unit.

Note: Since analog outputs cannot be used, DX120/DX121 cannot be used.

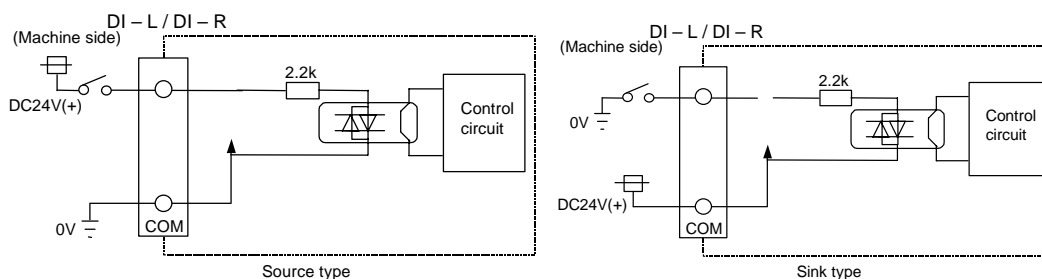
18 Machine Support Functions

18.1 PLC

(2) Outline of digital signal input circuit

There is a sink type and source type digital signal input circuit. The type is selected with a card unit in each unit.

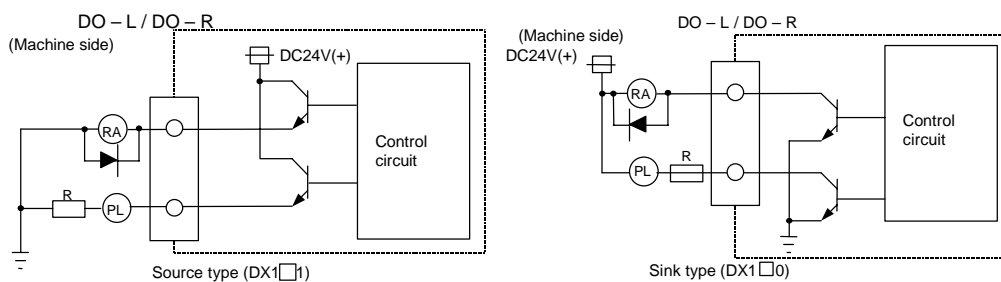
Input circuit



(3) Outline of digital signal output circuit

There is a sink type (DX1□0) and source type (DX1□1) digital signal output circuit. Use within the range of the specifications given below.

Output circuit



Output conditions

Insulation method	Non-insulated
Rated load voltage	DC+24V
Max. output current	60 mA
Output delay time	40 μs

<Caution>

- * When using an inductive load such as a relay, always connect a diode (withstand voltage 100V or more, 100mA or more) in parallel with the load. The diode should be inserted as close to the load (within 20cm) as possible.
- * When using a capacitive load such as a lamp, connect a protective resistor (R=150 Ω) in serial with the load to limit the rush current. (Make sure that the current is lower than the above tolerable current, including momentary current.)

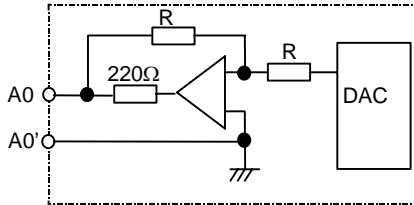
18 Machine Support Functions

18.1 PLC

(4) Outline of analog signal output circuit

The analog signal output circuit can be used only with the FCUA-DX120/DX121.

Output circuit



Output conditions

Output voltage	-10~ +10V (±5%)
Resolution	2^{12} (1/4095)
Load conditions	10 kΩ load resistance
Output impedance	220 Ω

(5) Input signal conditions

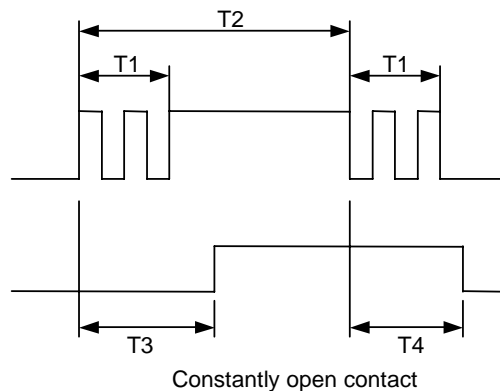
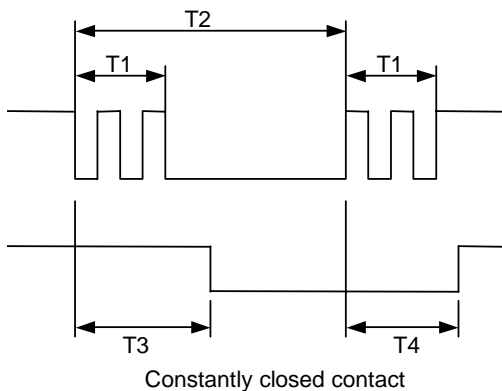
The input signals must be used within the ranges of the following conditions.

Source type <Contact common + 24V>

Input voltage at external contact ON	18 V to 25.2V
Input current at external contact ON	9 mA or more
Input voltage at external contact OFF	4 V or less
Input current at external contact OFF	2 mA or less
Tolerable chattering time	3 ms or less (Refer to T ₁ below)
Input signal hold time	40 ms or more (Refer to T ₂ below)
Input circuit operation delay time	$3 \text{ ms} \leq T_3 \cong T_4 \leq 20 \text{ ms}$
Machine side contact capacity	30 V or more, 16 mA or more

Sink type <Contact common grounding (RG)>

Input voltage at external contact ON	6 V or less
Input current at external contact ON	9 mA or more
Input voltage at external contact OFF	20 V to 25.2V
Input current at external contact OFF	2 mA or less
Tolerable chattering time	3 ms or less (Refer to T ₁ below)
Input signal hold time	40 ms or more (Refer to T ₂ below)
Input circuit operation delay time	$3 \text{ ms} \leq T_3 \cong T_4 \leq 20 \text{ ms}$
Machine side contact capacity	30V or more, 16 mA or more



18 Machine Support Functions

18.1 PLC

18.1.5 Ladder Monitor

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

This function enables the operating status of the sequence circuit to be checked on the controller's setting and display unit.

The monitor functions include the following.

- (1) Circuit monitoring
- (2) Screen stop by monitor stop trigger point
- (3) Entry monitoring
- (4) Decimal-hexadecimal conversion present value monitoring

18.1.6 PLC Development

18.1.6.1 On-board Development

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

PLC ladders can be developed on the control unit.

PLC ladder circuits can be created, edited, etc.

18.1.6.2 MELSEC Development Tool

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

The GX-Developer installed in a personal computer (OS:Windows) can be used.

18.1.6.3 PLC4B

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

This is the conventional PLC development tool for MELDAS.

18.1.7 C Language Function

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	△	△	-	-	△	△	△	△
L system	△	△	-	-	△	△	△	△

PLC subprograms prepared in C language can be called from PLC ladders.

18 Machine Support Functions

18.1 PLC

18.1.8 PLC Protect

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	—	—	—	—	—	—
L system	Δ	Δ	—	—	—	—	—	—

The user PLC protect function is configured as follows.

(1) PLC protect

The dedicated option protect release cassette must be installed in CBUS#1 if the following operations are to be performed when the PLC protect option is enabled. (For CBUS#1 and CBUS#2, refer to the figure below.)

<Operations requiring option protect release cassette>

- PLC editing operations using on-board editing tool (write, insert, delete, change)
- Ladder input/output

(2) Version upgrade cassette preparation

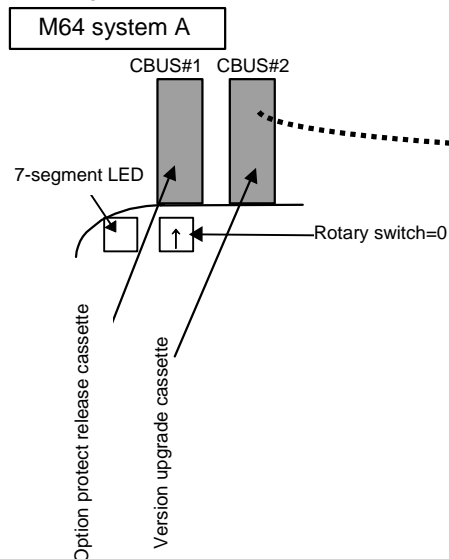
The version upgrade cassette is prepared by writing the M64 format ladders into the ROM cassette. The versions of the ladders for other M64 systems can then be upgraded using this cassette.

(3) PLC version upgrading

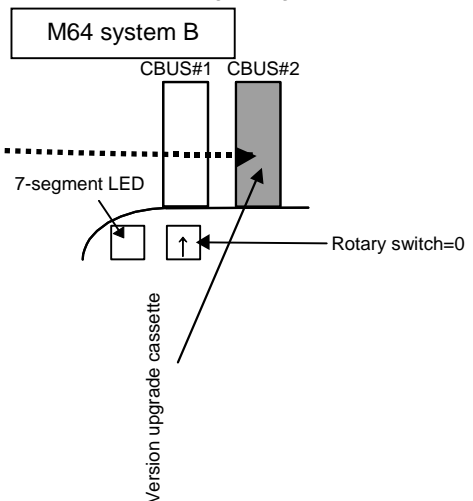
By installing the version upgrade cassette in CBUS#2 and turning the power back on, the PLC ladders inside the version upgrade cassette are transferred inside the CNC system and written into the F-ROM.

Example: Transplanting the PLC ladders of system A to system B

Version upgrade cassette preparation



PLC version upgrading



18 Machine Support Functions

18.1 PLC

18.1.9 PLC Password Lock

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ

This function makes it possible to use a code number to prohibit user PLC editing and input/output in order to prevent the illegal writing by the end users of the user PLC data prepared by the machine tool builder.

This function can be used when the ladder type is the PLC4B type.

User PLC protection using code number

On-board PLC file writing, editing operations (write, insert, delete, change) for PLC circuits and PLC file input/output operations are enabled by inputting the code number.

The operations which are prohibited during user PLC protection by the code number are listed in the table below.

List of operations

Operation		Code number input	No code number input	Remarks
Ladder circuits	Readout	Possible	Possible	
	Write	Possible	Impossible	
	Insert	Possible	Impossible	
	Delete	Possible	Impossible	
	Change	Possible	Impossible	
	Monitor	Possible	Possible	
PLC files	RUN/STOP	Possible	Possible	
	Write	Possible	Impossible	
	Input/output	Possible	Impossible	
External alarm messages	Input/output	Possible	Impossible	

18 Machine Support Functions

18.1 PLC

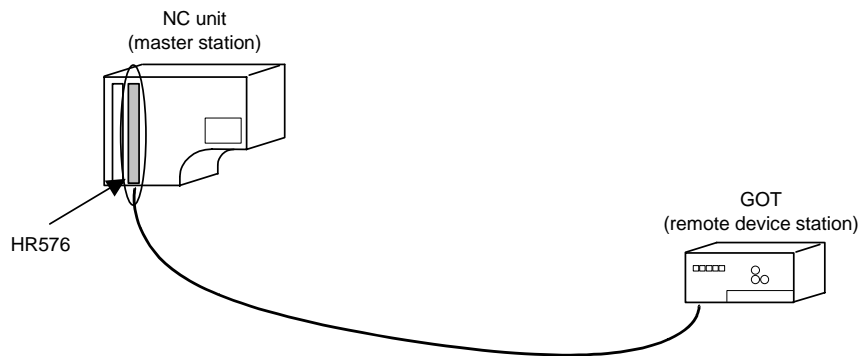
18.1.12 GOT Connection

18.1.12.2 CC-Link Connection (Remote Device)

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	Δ	Δ	Δ	Δ
L system	-	-	-	-	Δ	Δ	Δ	Δ

The GOT can be connected as a CC-Link remote device.

Note that the devices that can be used with GOT are limited as remote devices.



18 Machine Support Functions

18.1 PLC

18.1.13 PLC Message

18.1.13.1 Japanese

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
	M system	○	○	○	○	○	○	○	○
	L system	○	○	○	○	○	○	○	○

18.1.13.2 English

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
	M system	○	○	○	○	○	○	○	○
	L system	○	○	○	○	○	○	○	○

18.1.13.3 German

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
	M system	○	○	○	○	○	○	○	○
	L system	○	○	○	○	○	○	○	○

18.1.13.4 Italian

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
	M system	○	○	○	○	○	○	○	○
	L system	○	○	○	○	○	○	○	○

18.1.13.5 French

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
	M system	○	○	○	○	○	○	○	○
	L system	○	○	○	○	○	○	○	○

18.1.13.6 Spanish

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
	M system	○	○	○	○	○	○	○	○
	L system	○	○	○	○	○	○	○	○

18.1.13.9 Portuguese

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
	M system	○	○	○	○	○	○	○	○
	L system	○	○	○	○	○	○	○	○

18 Machine Support Functions

18.1 PLC

18.1.13.10 Hungarian

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

18.1.13.11 Dutch

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

18.1.13.12 Swedish

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

18 Machine Support Functions

18.2 Machine Construction

18.2 Machine Construction

18.2.1 Servo OFF

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
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 for some reason or other 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 is designated, the movement amount will be compensated even in the emergency stop state.

The axis is simultaneously set with servo OFF to 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 value display will not deviate.

18 Machine Support Functions

18.2 Machine Construction

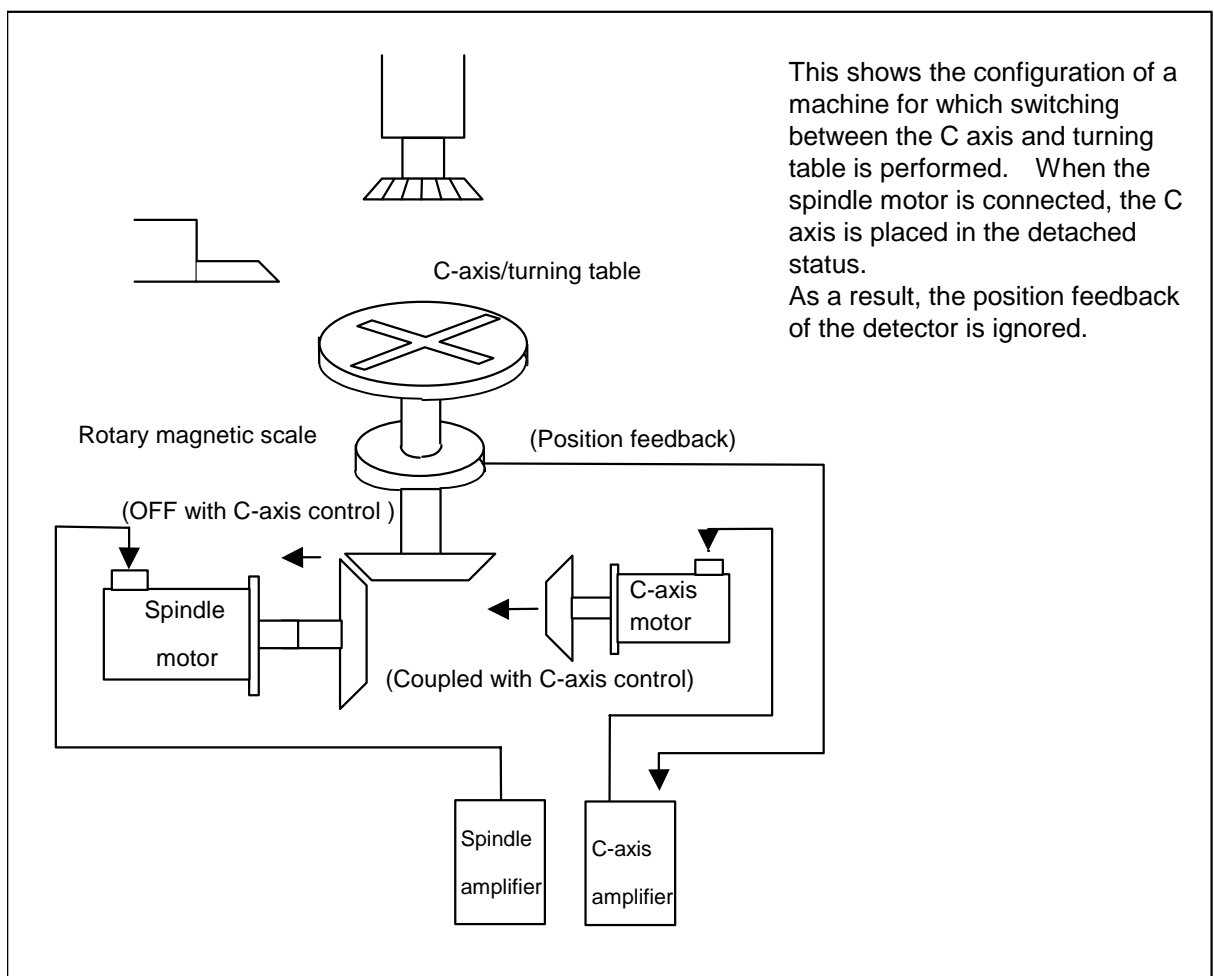
18.2.2 Axis Detach

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ

This function enables the control axis to be freed from control. Conversely, an axis which has been freed from control can be returned to the control status.

This function enables the rotary table or attachments to be removed and replaced.

Automatic operation is disabled until the axis for which the axis detach command has been released completes its dog-type reference position return.



```

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 current value display on the POSITION screen and at the same time the servo ready for the controller output signal is set to OFF. The current 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.

18 Machine Support Functions

18.2 Machine Construction

18.2.3 Synchronous Control

18.2.3.1 Position Tandem

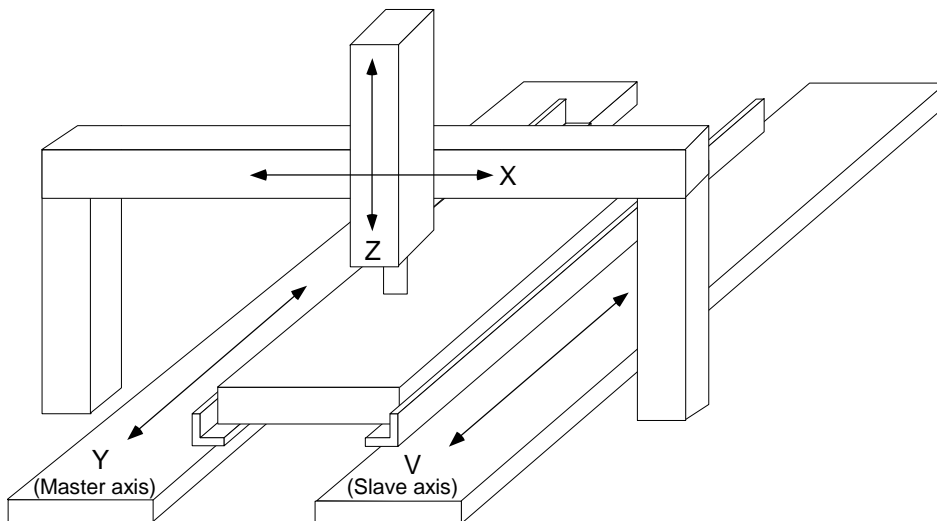
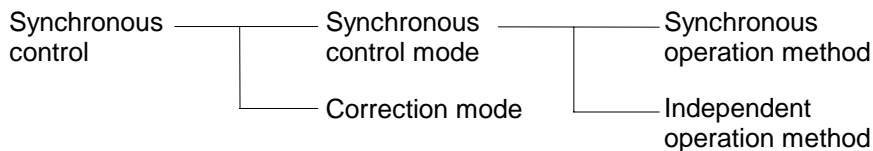
M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	Δ	Δ	Δ	-	Δ	Δ	Δ
L system	-	-	-	-	-	-	-	-

The synchronous control is a control method that both master and slave axes are controlled with the same movement command by designating the movement command for the master axis also to the slave axis. This function is assumed to be used in the large machine tool, etc. which drives one axis with two servo motors.

The axis 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 can not 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.
- Two sets are applied for the master and slave axes



18 Machine Support Functions

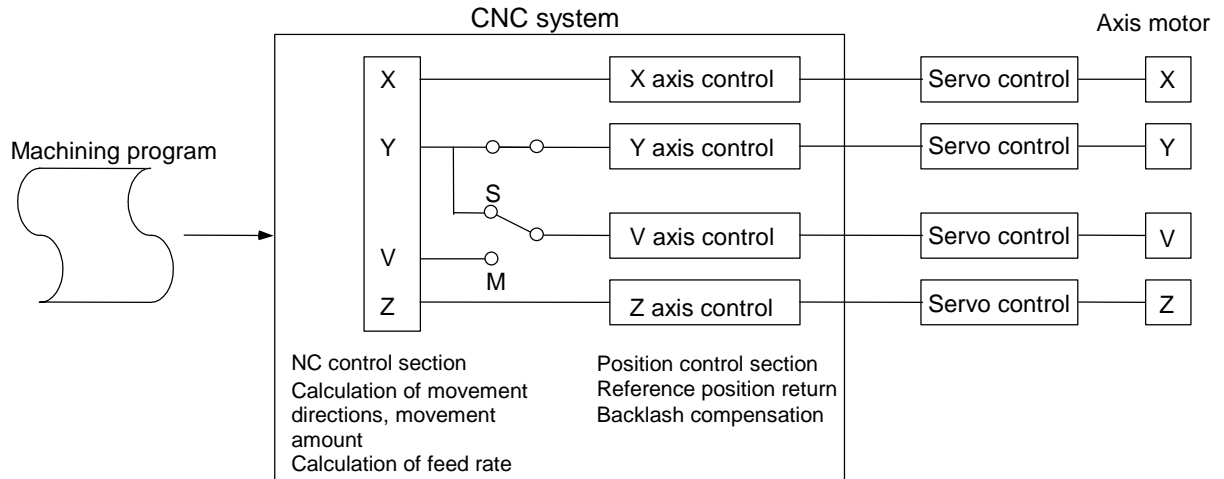
18.2 Machine Construction

(1) Synchronous control mode

The following two operation methods are available in the synchronous control mode.

(a) Synchronous operation

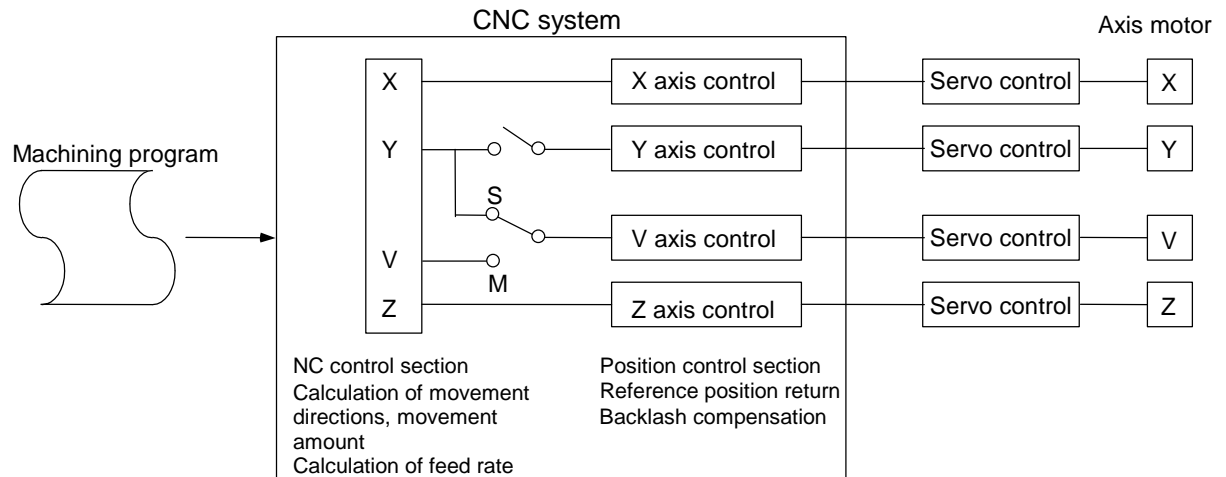
This is a method that both master and slave axes are moved simultaneously with the movement command for the master axis.



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 allowable synchronization error value set in the parameter is exceeded. However, when the zero point is not established, the synchronous error is not checked.

(b) Independent operation

This is a method that either the master or slave axis is moved with the movement command for the master axis.



(2) Correction mode

The synchronization is temporarily 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.

18 Machine Support Functions

18.2 Machine Construction

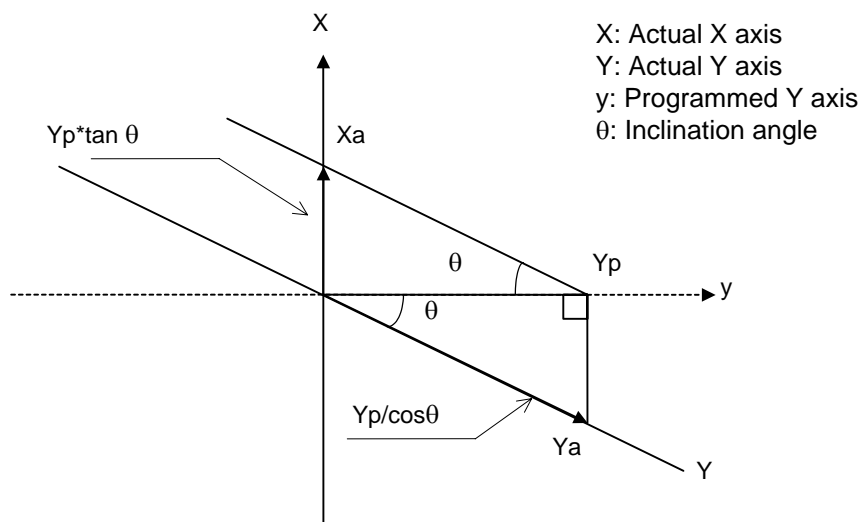
18.2.4 Inclined Axis Control

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	-	-	-	-
L system	-	Δ (*)	Δ (*)	Δ (*)	Δ	Δ	Δ	Δ

Even when the control axes configuring that machine are mounted at an angle other than 90 degrees, this function enables it to be programmed and controlled in the same way as with 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 offset using this angle.

<Example of use> When the X axis serves as the basic axis and the Y axis serves as the inclined axis axis



The Y-axis position and Y_p on the programmed coordinates (on the orthogonal coordinates) are respectively the X_a and Y_a positions produced by combining the X axis and Y axis along which the machine actually moves.

Therefore, the Y-axis (inclined axis) movement amount is expressed by the following formula:

$$Y_a = Y_p / \cos\theta \quad (1)$$

The X-axis (basic axis) movement amount is offset by the inclined movement of the Y axis, and it is expressed as follows:

$$X_a = X_p - Y_p \times \tan\theta \quad (2)$$

The Y-axis (inclined axis) speed is as follows:

$$F_a = F_p / \cos\theta$$

X_a , Y_a and F_a are the actual movement amounts and speed.
 X_p , Y_p and F_p are the movement amounts and speed on the program coordinates.

(Note) (*) indicates that X axis (fixed) is used as the inclined axis.

18 Machine Support Functions

18.2 Machine Construction

18.2.5 Index Table Indexing

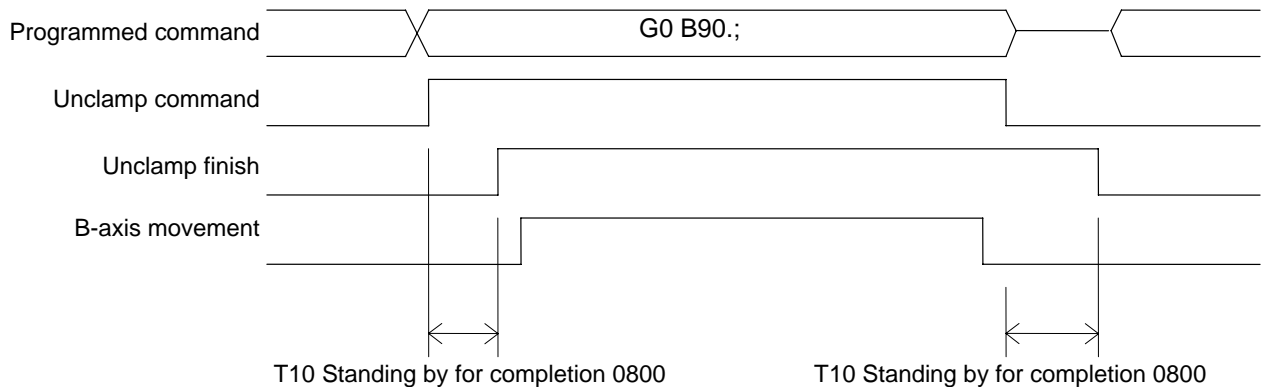
M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

The indexing of the index table can be performed by setting the index axes. Programming is facilitated because, in terms of the index commands, only the indexing angle need to be designated using the address of the programmed axis serving as the index setting axis, and there is no need to designate special M codes for clamping and unclamping the table.

The following operations are performed for the index table indexing function.

- (1) The "index axis selection" parameter (#2076) for the axis along which the indexing table is to be indexed is set to "1."
- (2) Designate the programmed commands (absolute or incremental) for moving the selected axis.
- (3) The unclamp command signal is now output prior to the axis movement.
- (4) When the axes are unclamped, the unclamp finish signal is set (ladder used for processing).
- (5) After checking the unclamp finish signal, the designated axis starts moving.
- (6) Upon completion of the movement, the unclamp command signal is set to OFF.
- (7) Clamp the axes and set the unclamp finish signal to OFF (ladder used for processing).
- (8) After checking that the unclamp finish signal is OFF, processing of the next block is initiated.

Operation timing chart



18 Machine Support Functions

18.2 Machine Construction

18.2.6 NSK Table Connection Control

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ

By assigning commands to the control unit from the 2nd miscellaneous function and the PLC and pulse train input, this function transmits commands by serial communication (RS-232C specification) to the dedicated drive unit of the NSK mega-torque motor.

Command designation method

By setting the command for the prescribed R register and the numerical value command following address B designated by the 2nd miscellaneous function as signed binary numbers using the PLC, the control unit prepares the positioning commands from this command and numerical value command, and it sends them to the drive unit by serial communication (RS-232C specification). Furthermore, by setting ON the handle mode of the prescribed R register using the PLC, the pulse train input based on the handle is sent as the movement command to the drive unit. Designate the commands with the number of mega-torque motor axes connected already set in the parameter and with the settings for the input/output basic parameters used for communication already set. Up to 16 axes can be connected as the mega-torque motor axes.

The numerical value command in angle increments is prepared by setting the numerical value following address B of the 2nd miscellaneous function as a signed binary number in the prescribed R register using the PLC.

18 Machine Support Functions

18.2 Machine Construction

18.2.7 Auxiliary Axis Control

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ

The MR-J2-CT drive unit for positioning and indexing can be connected for auxiliary axis control. The drive unit is a single-axis control unit, and the control is performed from the PLC. It comes with the following functions, and is suited to controlling a peripheral device of the machine.

- (1) Feed functions
 - (a) Four different feed rates can be set and selected using parameter settings.
 - (b) Constant inclination acceleration/deceleration, linear acceleration/deceleration or soft acceleration/deceleration can be selected.
 - (c) When rotary axis is used, automatic short-cut discrimination and rotary direction can be assigned by commands.

- (2) Command methods
 - (a) Station method
Any point (station) obtained when the rotary axis has been divided into equal parts can be selected by a command, and the axis can be positioned at that point. The maximum number of divisions is 360.
 - (b) Arbitrary coordinate designation method
The arbitrary coordinates (absolute position as referenced to the zero point) can be commanded from the PLC and the axis can be positioned at these coordinates.

- (3) Operation functions
 - (a) JOG mode
In this mode, the axis is rotated at a constant speed in the designated direction while the start signal is ON.
 - (b) Automatic mode
In this mode, the axis is positioned at the designated station number by the start signal.
 - (c) Manual mode
In this mode, the axis is rotated at a constant speed in the designated direction while the start signal is ON. When the start signal is set to OFF, the axis is positioned at the nearest station position.
 - (d) Arbitrary coordinate mode
In this mode, the axis is positioned at the arbitrary coordinates designated with the PLC by the start signal. When the start signal is set to OFF prior to the completion of the positioning, the axis immediately decelerates and stops.
 - (e) Manual handle mode
In this mode, axis travel is carried out by the pulse command (manual handle command) sent from the PLC.
 - (f) Reference position return mode
In this mode, the axis is positioned at the coordinate reference position. Two methods are used: one method is based on a dog switch and the other method is to carry out positioning to the reference position which is stored in the memory.
 - (g) Press-fit-and-positioning mode
In this mode, the axis is positioned while it is pressed against the machine end, etc.

18 Machine Support Functions

18.3 PLC Operation

18.3 PLC Operation

18.3.1 Arbitrary Feed in Manual Mode

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
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).

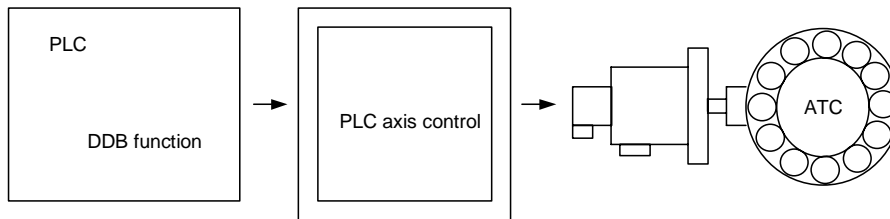
18 Machine Support Functions

18.3 PLC Operation

18.3.3 PLC Axis Control

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ

Over and above the NC control axes, this function enables axes to be controlled independently by commands based on the PLC.



Item	Description
Number of control axes	Max. 2 axes
Simultaneously controlled axes	PLC control axis is controlled independently from NC control axes. A multiple number of PLC axes can be started simultaneously.
Command increment	Least command increment 0.001mm (0.0001 inch) 0.0001mm (0.00001 inch) (Same as command increment for NC control axes)
Feed rate	Least command increment: 0.001mm Rapid traverse 0 to 1000000 mm/min (0 to 100000 inch/min) Cutting feed 0 to 1000000 mm/min (0 to 100000 inch/min) Least command increment: 0.0001mm Rapid traverse 0 to 100000 mm/min (0 to 10000 inch/min) Cutting feed 0 to 100000 mm/min (0 to 10000 inch/min)
Movement commands	Incremental commands from current position Absolute commands for machine coordinate system 0 to ±99999999 (0.001mm/0.0001 inch)
Operation modes	Rapid traverse, cutting feed, jog feed (+) (-), reference position return feed (+) (-), handle feed
Acceleration/deceleration	Rapid traverse, jog feed, reference position return feed Linear acceleration/deceleration Cutting feed Exponential function acceleration/deceleration Handle feed Step
Backlash compensation	Available
Stroke end	None
Soft limit	Available
Rotary axis command	Available For absolute commands: amount within 1 rotation (rotation by amount remaining after division into 360) For incremental commands: rotation by assigned amount
Inch/mm changeover	None Set to the command that corresponds to the feedback unit.
Position detector	Encoder (Absolute position can also be detected.)

18 Machine Support Functions

18.4 PLC Interface

18.4 PLC Interface

18.4.1 CNC Control Signal

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

Control commands to the CNC system are assigned from the PLC. Input signals with an A/D conversion function and skip inputs that respond at high speed can also be used.

(1) Control signals

- Control signals for operations in automatic operation mode
- Control signals for operations in manual operation mode
- Control signals for program execution
- Control signals for interrupt operations
- Control signals for servo
- Control signals for spindle
- Control signals for mode selection
- Control signals for axis selection
- Control signals for feed rates

(2) Analog voltage control [M system]

When an analog voltage is input to an external connector used to connect CNC analog inputs, the data corresponding to the input voltage can be read out in the prescribed file register. This data can be used for load meter displays, thermal deformation compensation, etc. (Maximum 8 points)

(3) Skip signals

When signals are input to the skip input interface, they are processed by interrupt processing. This enables functions requiring a high response speed to be implemented. (Maximum 4 points)

For further details, refer to the PLC Interface Manual.

18 Machine Support Functions

18.4 PLC Interface

18.4.2 CNC Status Signal

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	○	○	○	○	○	○	○	○
L system	○	○	○	○	○	○	○	○

The status signals are output from the CNC system. They can be utilized by referencing them from the PLC.

These signals can also be output as analog data by setting the data from the PLC in the R register.

Status output functions

(1) Controller operation ready

When the controller power is turned ON and the controller enters the operation ready status, the "Ready" signal is output to the machine.

Refer to the PLC Interface Manual for details of the sequences from when the controller power is supplied to when the controller ready status is entered.

(2) Servo operation ready

When the controller power is turned ON and the servo system enters the operation ready status, the "Servo ready" signal is output to the machine.

Refer to the PLC Interface Manual for details of the sequences from when the power is supplied to when the "Servo ready" signal is turned ON.

(3) In automatic operation

Generally, if the "cycle start" switch is turned ON in the automatic operation mode (memory, MDI), this signal is output until the reset state or emergency stop state is entered by the M02, M30 execution or the reset & rewind input to the controller using the reset button.

(4) In automatic start

The signal that denotes that the controller is operating in the automatic mode is output from the time when the cycle start button is pressed in the memory or MDI mode and the automatic start status has been entered until the time when the automatic operation is terminated in the automatic operation pause status entered by the "feed hold" function, block completion stop entered by the block stop function or resetting.

(5) In automatic pause

An automatic operation pause occurs and this signal is output during automatic operation from when the automatic pause switch is pressed ON until the automatic start switch is pressed ON, or during automatic operation when the mode select switch is changed from the automatic mode to the manual mode.

(6) In rapid traverse

The "In rapid traverse" signal is output when the command now being executed is moving an axis by rapid traverse during automatic operation.

(7) In cutting feed

The "In cutting feed" signal is output when the command now being executed is moving an axis by cutting feed during automatic operation.

(8) In tapping

The "In tapping" signal is output when the command now being executed is in a tap modal which means that one of the statuses below is entered during automatic operation.

- (a) G84 (fixed cycle: tapping cycle)
- (b) G74 (fixed cycle: reverse tapping cycle)
- (c) G63 (tapping mode)

18 Machine Support Functions

18.4 PLC Interface

(9) In thread cutting

The "In thread cutting" signal is output when the command now being executed is moving an axis by thread cutting feed during automatic operation.

(10) In rewinding

The "In rewinding" signal is output when the reset & rewind signal is input by M02/M30, etc., during memory operation and the program currently being executed is being indexed.

The rewinding time is short, so there may be cases when it cannot be confirmed with the sequence program (ladder).

(11) Axis selection output

The "Axis selection output" signal for each axis is output to the machine during machine axis movement.

(a) Automatic mode

The signal is output in the movement command of each axis. It is output until the machine stops during stop based on feed hold or block stop.

(b) Manual mode (including incremental feed)

The signal is output while the axis is moving from the time when the jog feed signal is turned ON until the time when it is turned OFF and the machine feed stops.

(c) Handle feed mode

The signal is output at all times when the axis selection input is on.

(12) Axis movement direction

This output signal denotes the direction of the axis now moving, and for each axis a "+" (plus) signal and a "-" (minus) signal are output respectively.

(13) Alarm

This signal indicates the various alarm statuses that arise during controller operation. It is divided into the following types and output.

(a) System errors

(b) Servo alarms

(c) Program errors

(d) Operation errors

(14) In resetting

The "Reset" signal is output during the reset process when the reset & rewind command is input to the controller with the "reset" button on the setting and display unit is pressed or when the "Reset" signal is input from the machine operation panel, etc.

This signal will also be output when the controller READY status is OFF, when the Emergency stop signal is input or when a servo alarm is occurring, etc.

(15) Movement command finish

In the memory or MDI automatic operation, the "Movement command finish" signal is output when the command block in the machining program features a movement command and when that block command has been completed.

When the movement command and M, S, T or B command have been assigned in the same block, then the movement command signal can be used as a sync signal for either executing the processing of the M, S, T or B command at the same time as the command or executing it upon completion of the movement command.

18 Machine Support Functions

18.4 PLC Interface

18.4.5 DDB

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ

The DDB (direct data bus) provides the function for PLC to directly read/write controller data. PLC can read the specified data into a buffer and set (write) the specified data into the controller by setting information required for read/write in the buffer and calling the DDB function. Generally, data is read/written for each data piece, but data related to control axes is processed in batch for as many axes as the specified number of axes.

The feature of the DDB function is the capability of referencing read data or write data in the next step just after a DDBA instruction is executed.

18 Machine Support Functions

18.5 Machine Contact I / O

18.5 Machine Contact I / O

DI:48/DO:48

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
M system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
L system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

DI:64/DO:64

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
M system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
L system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

Additional DI/DO (DI:64/DO:48)

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
M system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
L system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

Additional DI/DO(DI:32/DO:32)

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
M system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
L system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

Operation board IO (DI:32/DO:32)

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
M system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
L system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

Operation board IO DI:64/DO:48

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
M system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
L system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

Remote IO 32/32

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
M system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
L system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

Remote IO 64/48

Model		M64/65/66				M64/65/66S			
		M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M/L system									
M system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
L system		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

18 Machine Support Functions

18.6 External PLC Link

18.6 External PLC Link

18.6.1 External PLC link I (M-NET)

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	-	-	-	-
L system	Δ	Δ	Δ	Δ	-	-	-	-

This function enables a serial link to be established using RS-422 as another machine interface in addition to the existing remote I/O.

In addition to the 9.6, 19.2 and 38.4 kbps baud rates, baud rates of 14.4, 28.8 and 57.5 kbps are supported.

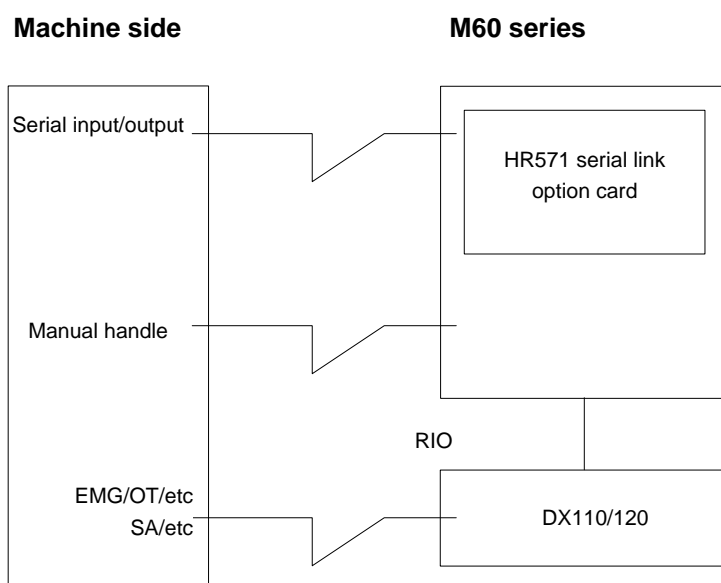
This function enables the serial link to be established by adding the HR571 dedicated interface card.

Basic specifications of serial link

Item	Basic specification
Number of input/output points	Max. number of inputs: 256 points (max. 128 points per station when MELSEC is connected) Max. number of outputs: 256 points (max. 128 points per station when MELSEC is connected) (Inputs and outputs can be set in 8-point increments.)
Serial connection	RS-422 standard satisfied, 1 channel
Baud rates	9.6, 19.2 and 38.4 kbps/14.4, 28.8 and 57.5 kbps
Connecting sequencer	MELSEC-A (A1CPU, A2CPU, A3CPU) + AJ71C22/AJ71C22S1 (max. 38.4 kbps) or equivalent product made by another company
Transmission specifications	Refer to AJ71C22 transmission specifications.

Connections between the equipment

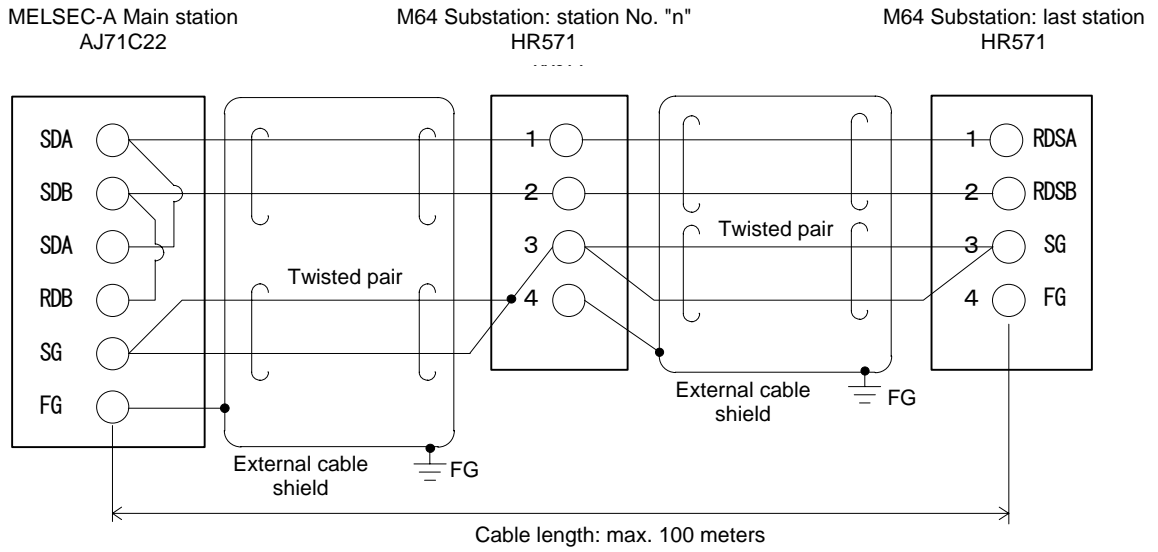
The following figure shows how the equipment is connected in a serial connection format.



18 Machine Support Functions

18.6 External PLC Link

Cable connections



18 Machine Support Functions

18.6 External PLC Link

18.6.2 External PLC Link II (Bus connection)

18.6.2.1 MELSEC-A, QnA Series

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ

Connection with an external PLC (Mitsubishi MELSEC-A/QnA Series) is possible with a bus.

Item	Basic specifications
Number of input/output points	Bit data input: max. 256 points, output: max. 256 points
	Numerical data input: max. 96 words, output: max. 128 words
	Combined use with remote I/O unit is possible.
Connection sequence	MELSEC-A/QnA Series
Number of connectable units	A maximum of four NC units can be connected to one MELSEC unit.

(Note 1) A maximum of four units can be connected, but the total length should be 6m or less.

(Note 2) The Number of NC units that can be connected will differ according the max No. of input/output points on the MELSEC-CPU.

(Note 3) The input signals for the hardware OT and reference position return near-point detection are connected to the remote I/O unit.

- (1) Software process
 - The signal interface is determined between the MELDAS and MELSEC.
 - The MELDAS side executes sequence processing with the built-in PLC to the interface exclusive for bus connection.
 - After reading in the data with the FROM command and processing the sequence, the MELSEC side will send the data to MELDAS with the TO command.
- (2) List of CPUs that can be bus connected to MELSEC
(One NC unit is regarded as one special unit.)

MELSEC CPU	Max. No. of IO points	Basic base	No. of expandable steps
A1NCPU	256 points	A**B	4 steps (Note)
A1SCPU		A1S**B	1 step
A1SJCPU			
A2NCPU	512 points	A**B	4 steps (Note)
A2ACPU			
A2UCPU		A1S**B	1 step
A2USCPU			
A2USHCPU-S1	1024 points	A1S**B	1 step
A2NCPU-S1		A**B	4 steps (Note)
A2ACPU-S1			
A2UCPU-S1			
A3NCPU	2048 points	A**B	4 steps (Note)
A3ACPU			
A3UCPU			
A4USPU			

(Note) The number of expandable steps, which is 7 steps when using only MELSEC, is limited to 4 units (4 steps) that is the number of connectable units for MELDAS.

MELSEC CPU	Max. No. of IO points	Basic base	No. of expandable steps
Q2ACPU	512 points	A**B	4 steps
Q2ACPU-S1	1024 points		
Q3ACPU	2048 points		
Q4ACPU	4096 points		

18 Machine Support Functions

18.6 External PLC Link

18.6.3 IO Link

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ
L system	Δ	Δ	Δ	Δ	Δ	Δ	Δ	Δ

Introduction

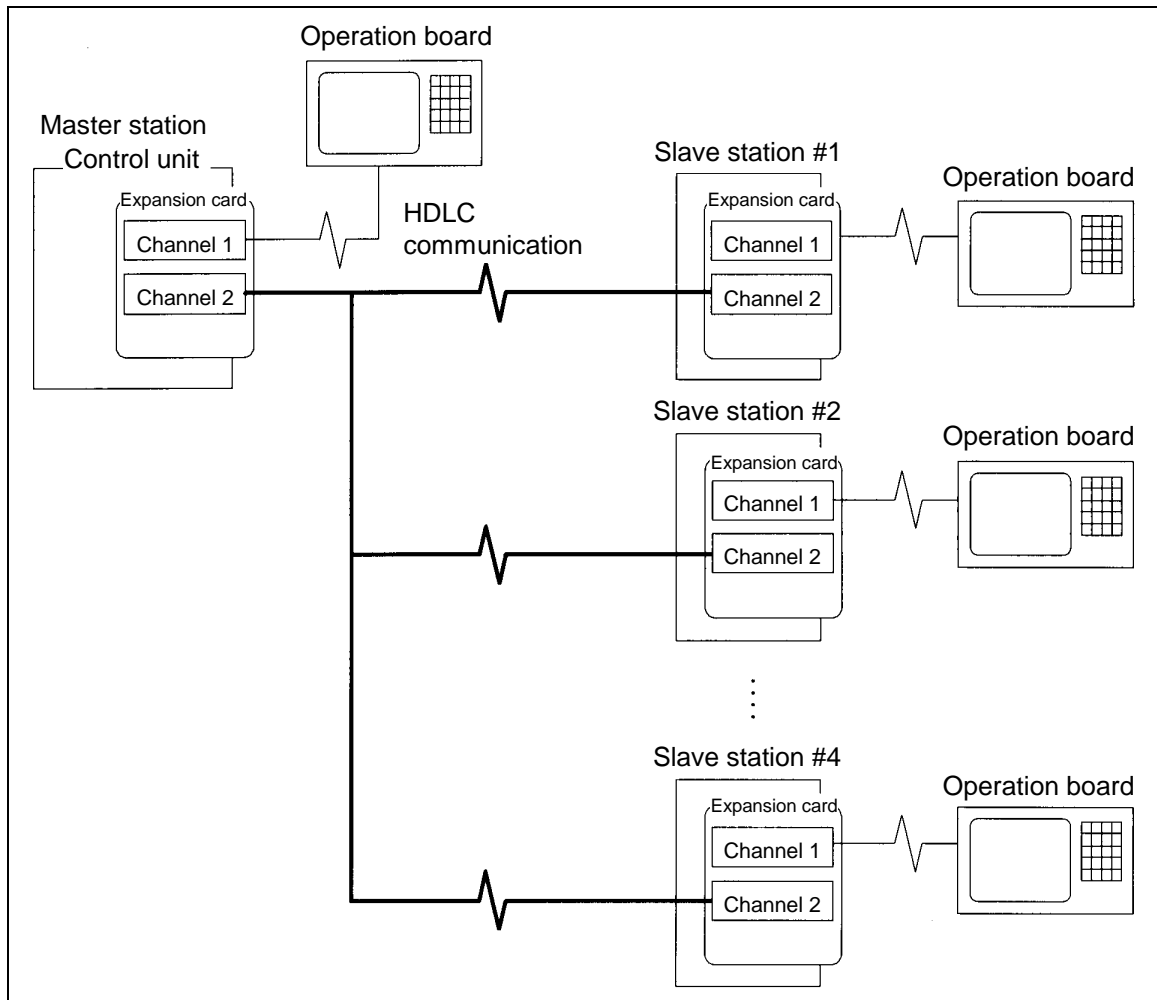
This function enables data to be sent and received between several numerical controllers by means of packet communications using the HDLC function.

It uses a different communication channel from the conventional communication between numerical controllers and operation board.

It enables communication between one master numerical NC and up to four slave NC stations.

System configuration

As shown in the figure below, the second channel of the expansion card in the control unit is used.



System configuration

Precautions

- (1) One master station must always be provided for these communications. Data cannot be communicated between slave stations.

18 Machine Support Functions

18.6 External PLC Link

18.6.4 CC-Link

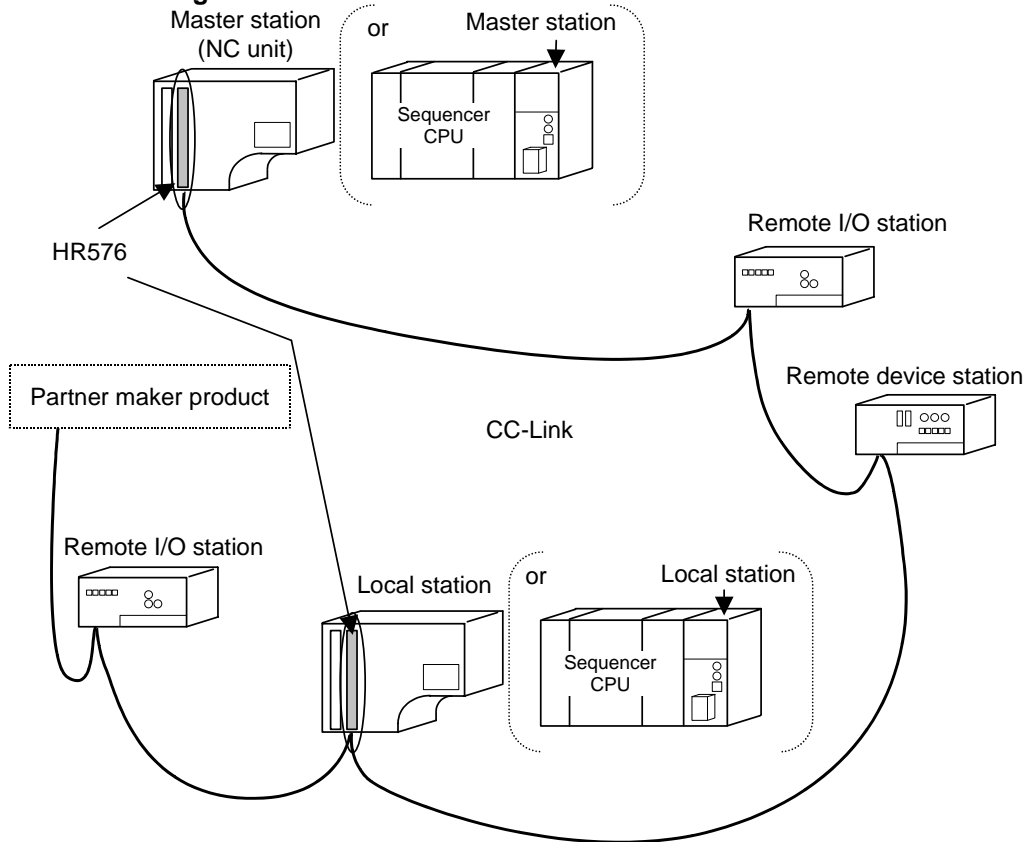
M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	-	-	-	-	Δ	Δ	Δ	Δ
L system	-	-	-	-	Δ	Δ	Δ	Δ

NC unit can be directly connected to the network to serve as the master/local station of the MELSEC CC-Link. To enable this connection, the CC-Link master/local units (HR576) must be installed in the expansion slots. When using this function, the user PLC ladder type must be the GX developer type. The transient command of the MELSEC A series using this function cannot be used. In connecting with the GOT, set the GOT as the remote device station. (Cannot set as intelligent station)

(1) Outline of CC-Link

- Distributing and installing each unit to the equipments such as conveyor line and mechanical device and so on can simplify the wiring of the whole system.
- The ON/OFF data and numerical data such as input/output treated by each unit can be communicated easily and at high speed.
- The simple distributed system can be established by connecting several sequencer CPUs or NCs.
- Connecting the device equipments made by the partner maker can flexibly support various systems.

(2) Outline drawing



Master station	This station controls the remote I/O station, remote device station and local station.
Local station	This station has the CPU and can communicate with the master station and the other local station.
Remote I/O station	Remote station that treats only the ON/OFF data
Remote device station	Remote station that treats the ON/OFF data and numerical value

18 Machine Support Functions

18.6 External PLC Link

(3) Performance specifications

Item	CC-Link master/local unit (HR576)
Baud rates	156kbps/625kbps/2.5Mbps/5Mbps/10Mbps can be selected.
Max. transmission distance	The followings are obtained by the baud rate described above. 1200m/600m/200m/150m•110m/100m•80m•50m
Max. number of connection units	64 units Note that the following two conditions must be satisfied. Condition 1: $\{(1 \times a)+(2 \times b)+(3 \times c)+(4 \times d)\} \leq 64$ a: Number of units that occupy station 1 b: Number of units that occupy station 2 c: Number of units that occupy station 3 d: Number of units that occupy station 4 Condition 2: $\{(16 \times A)+(54 \times B)+(88 \times C)\} \leq 2304$ A: Number of remote I/O stations ≤ 64 units B: Number of remote device stations ≤ 42 units C: Local station, Standby master station, Number of intelligent device stations ≤ 26 units Number of stations
Number of occupied stations (Number of local stations)	Station 1 to station 4 (Changing over with DIP switch)
(Note 1) Max. number of link points per one system	Remote input/output (RX, RY) : Input/output each 2048 points Remote register (RWw) : 256 points (Master station → remote/local station) Remote register (RWr) : 256 points (Remote/local station → master station)
Number of link points per one remote station/local station	Remote input/output (RX, RY) : 32points (Local station is 30 points) (Note 2) Remote register (RWw) : 4 points (Mater station → remote/local station) Remote register (RWr) : 4 points (Remote/local station → master station)
Communication method	Polling method
Synchronization method	Flame synchronization method
Encode method	NRZI method
Transmission path method	Bus (RS485)
Transmission format	HDLC standard satisfied
Illegal control method	CRC ($X^{16} + X^{12} + X^5 + 1$)
Connection cable	Twist pair cable with shield
RAS function	<ul style="list-style-type: none"> • Automatic link refresh function • Sub-station isolation function • Link special relay/error detection by register
Number of Input/output occupied points	32 points

(Note 1) If the points which can be reserved as the device for the CC-Link in the NC side does not satisfied 2048 points, the number of points which can be reserved is the max. number of link points per one system.

The remote register (RWw, RWr) is also in the same manner.

(Note 2) The last station of each local station is 30 points.

Example: When the number of occupied station is the station 4,
Number of link points= $32 \times 3 + 30 = 126$ points

18 Machine Support Functions

18.6 External PLC Link

(4) Usable functions

In the CC-Link functions, the ones listed in the table below can be used by the NC.

Function item	MELSEC		NC unit	
	Master station	Local station	Master station	Local station
Communication between master station and remote I/O station	○	×	○	×
Communication between master station and remote device station	○	×	○	×
Communication between master station and local station	○	○	○	○
Mixed system communication	○	○	○	○
Reserved station function	○	×	○	×
Error cancel station function	○	×	○	×
Setting of data link status when trouble occurs in CPU of master station	○	○	○	○
Registration of parameters in EEPROM	○	×	× (Note 1)	× (Note 1)
Setting of input data status from data link trouble station	○	○	○	○
Unit resetting by sequence program	○	○	○	○
Data link stop/restart	○	○	○	○
LED diagnosis status	16-point display (A1SJ61QBT11)		16-point display	
Station number setting	Unit front panel switches		Setting switches on card	
Baud rate setting			Card front panel switches	
Mode setting switch				
Condition setting				
Automatic link refresh function	○	○	○	○
Sub-station isolation function	○	×	○	×
Data link status check (SB/SW)	○	○	○	○
Off-line test	○	○	○	○
On-line test	○	○	○	○
Monitor diagnosis	○	○	×	×
Parameter registration function	○	○	○ (Note 1)	○ (Note 1)
Automatic refresh function	○	○	○	○
Scan synchronization function	Synchronous mode	×	○	×
	Asynchronous mode	○	○	○
Standby master function (Note 2)	○	×	×	×
Temporary error cancel station designation function	○	×	○	×
Special-purpose instruction	○	○	Δ (Note 3)	Δ (Note 3)

(Note 1) The parameter for the CC-Link is set with the GX Developer.

When setting the CC-Link parameter, the format of the NC's ladder area is required in installing the HR576. Formatting is carried out with the GX Developer.

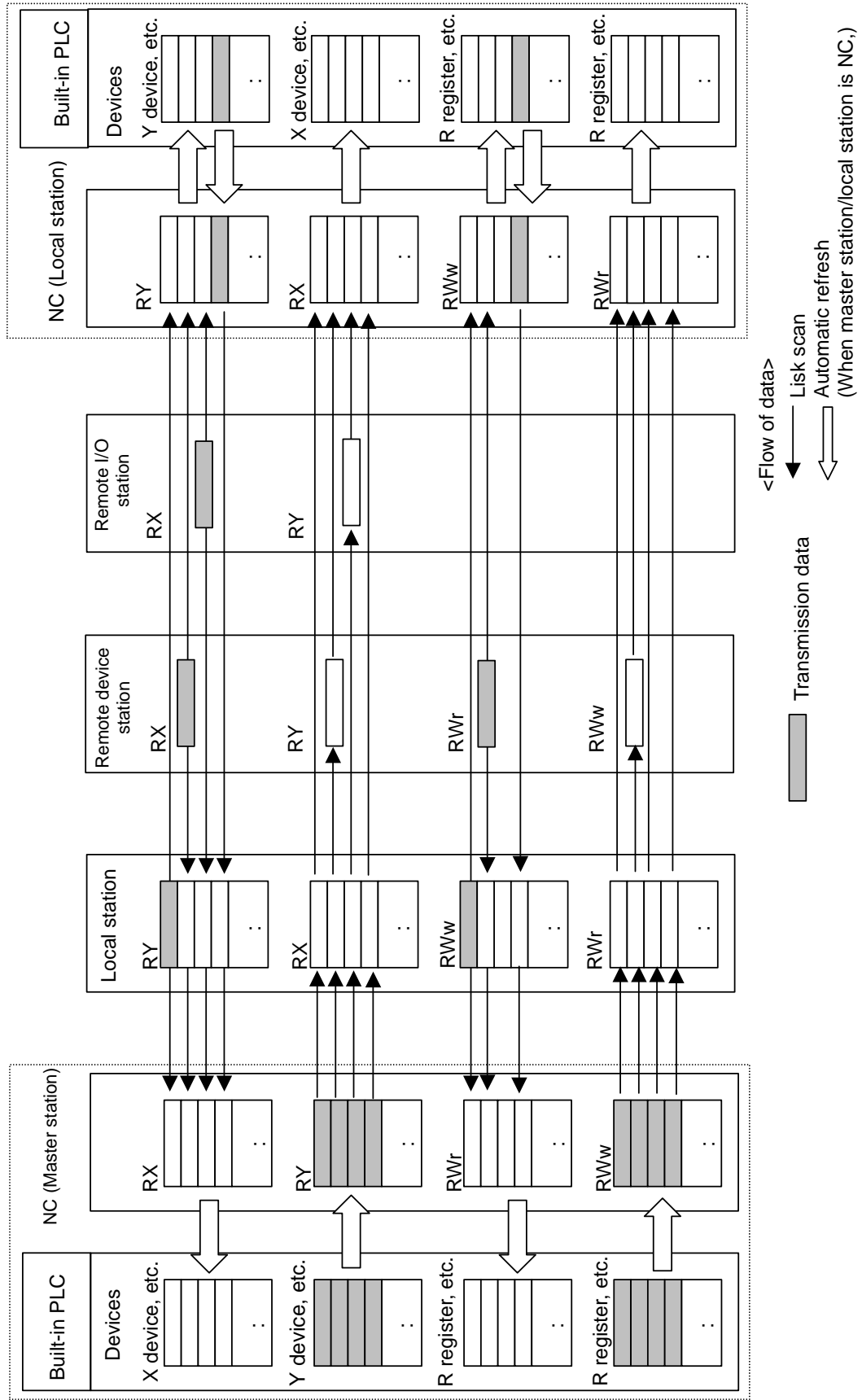
(Note 2) The special-purpose instruction (transient command) is corresponding to the RIRD command/RIWT command.

18 Machine Support Functions

18.6 External PLC Link

(5) Communication data flow

The flow of data communicated by the CC-Link's link scan is as follows.
 (The master station and local station of MELSEC CPU can be also mixed.)



18 Machine Support Functions

18.7 Installing S/W for Machine Tools

18.7 Installing S/W for Machine Tools

Software other than the built-in PLC can be installed in order to implement the machine tool builder's own functions (customized release). The customized release function consists of the following items.

- | | |
|--|---|
| (1) Screen release interface function | : Change of CNC standard screen, preparation of inherent screen |
| (2) DDB interface function | : Read/write CNC data |
| (3) Machine control interface function | : Set/reset PLC device |
| (4) File release interface function | : Preparation, modification, registration, etc. of user files using file system of CNC system |

18.7.1 APLC

M/L system \ Model	M64/65/66				M64/65/66S			
	M64A	M64	M65	M66	M64AS	M64S	M65S	M66S
M system	Δ	Δ	–	–	Δ	Δ	Δ	Δ
L system	Δ	Δ	–	–	Δ	Δ	Δ	Δ

The screens are released by pressing the "F0" function key (nothing is displayed on the screen of the NC unit). This enables the machine tool builder to display its own screens from its customized software.

Using the APLC libraries, the customized software enables screen displays (characters, graphics), key loading, file read/write, NC unit internal information read/write, and exchanges of R register and other information with PLC ladders.

Customized software is described using C language and developed using a commercial compiler.

Appendix 1. List of Specifications

Appendix 1. List of Specifications

Function code		Control unit recognition	Subject to parity V count	CRT display	Setting and display unit key-in	Stored in memory	Punch-out output		Internal NC system function
EIA	ISO						EIA	ISO	
0-9	0-9	Yes	Counted	Displayed	Key-in	Stored	0-9	0-9	Numerical data
A-Z	A-Z	Yes	Counted	Displayed	Key-in	Stored	A-Z	A-Z	Addresses
+	+	Yes	Counted	Displayed	Key-in	Stored	+	+	Sign, variable operator (+)
-	-	Yes	Counted	Displayed	Key-in	Stored	-	-	Sign, variable operator (-)
.	.	Yes	Counted	Displayed	Key-in	Stored	.	.	Decimal point
,	,	Yes	Counted	Displayed	Key-in	Stored	,	,	
/	/	Yes	Counted	Displayed	Key-in	Stored	/	/	Block delete (optional block skip), variable operator (÷)
EOR	%	Yes	Counted	Displayed (%)	No key-in (automatically inserted)	Stored	EOR	%	End of record (tape storage end), rewind start & stop during tape search
EOB/CR	LF/NL	Yes	Counted	Displayed (:)	Key-in, ;/EOB	Stored	EOB	LF	End of block
2+4+5	(Yes	Counted	Displayed	Key-in, ;/EOB	Stored	2+4+5	(Control out (comment start)
2+4+7)	Yes	Counted	Displayed	Key-in, ;/EOB	Stored	2+4+7)	Control in (comment end)
*	:	Yes	Counted	Displayed	No Key-in	Stored	*	:	Program number address (instead of O, ISO only)
*	#	Yes	Counted	Displayed	Key-in	Stored	*	#	Variable number
*	*	Yes	Counted	Displayed	Key-in	Stored	*	*	Variable operator (×)
*	=	Yes	Counted	Displayed	Key-in	Stored	*	=	Variable definition
*	[Yes	Counted	Displayed	Key-in	Stored	*	[Variable operator
*]	Yes	Counted	Displayed	Key-in	Stored	*]	Variable operator
BS	BS	No	Counted	Blank	No key-in	Stored			
TAB	HT	No	Counted	Blank	No key-in	Stored			
SP	SP	No	Counted	Blank	Key-in	Stored	SP (T-V automatic adjustment)	SP (T-V automatic adjustment)	SPs starting with EOB and ending when first character or number code appears are not subject to parity V count.
	CR	No	Counted	Blank	No key-in	Stored			
DEL	DEL	No	Not counted	Not displayed	No key-in	Not stored			
All space	NULL	No	Not counted	Not displayed	No key-in	Not stored			
All mark	(DEL)	No	Not counted	Not displayed	No key-in	Not stored			
Any other	Any other	No	Counted	(Note 3)	No key-in	Stored			

(Note 1) * indicates that corresponding code pattern can be set by parameter.

(Note 2) Codes not listed above are stored on tape but an error will result during operation if they are not comments.

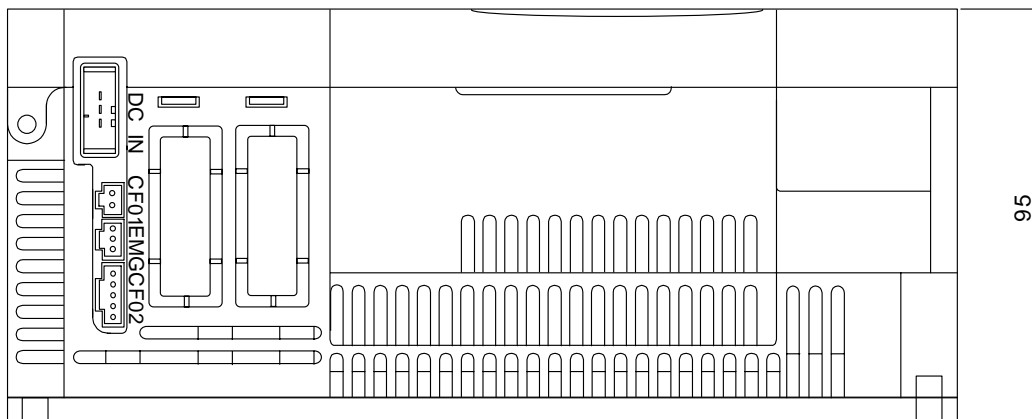
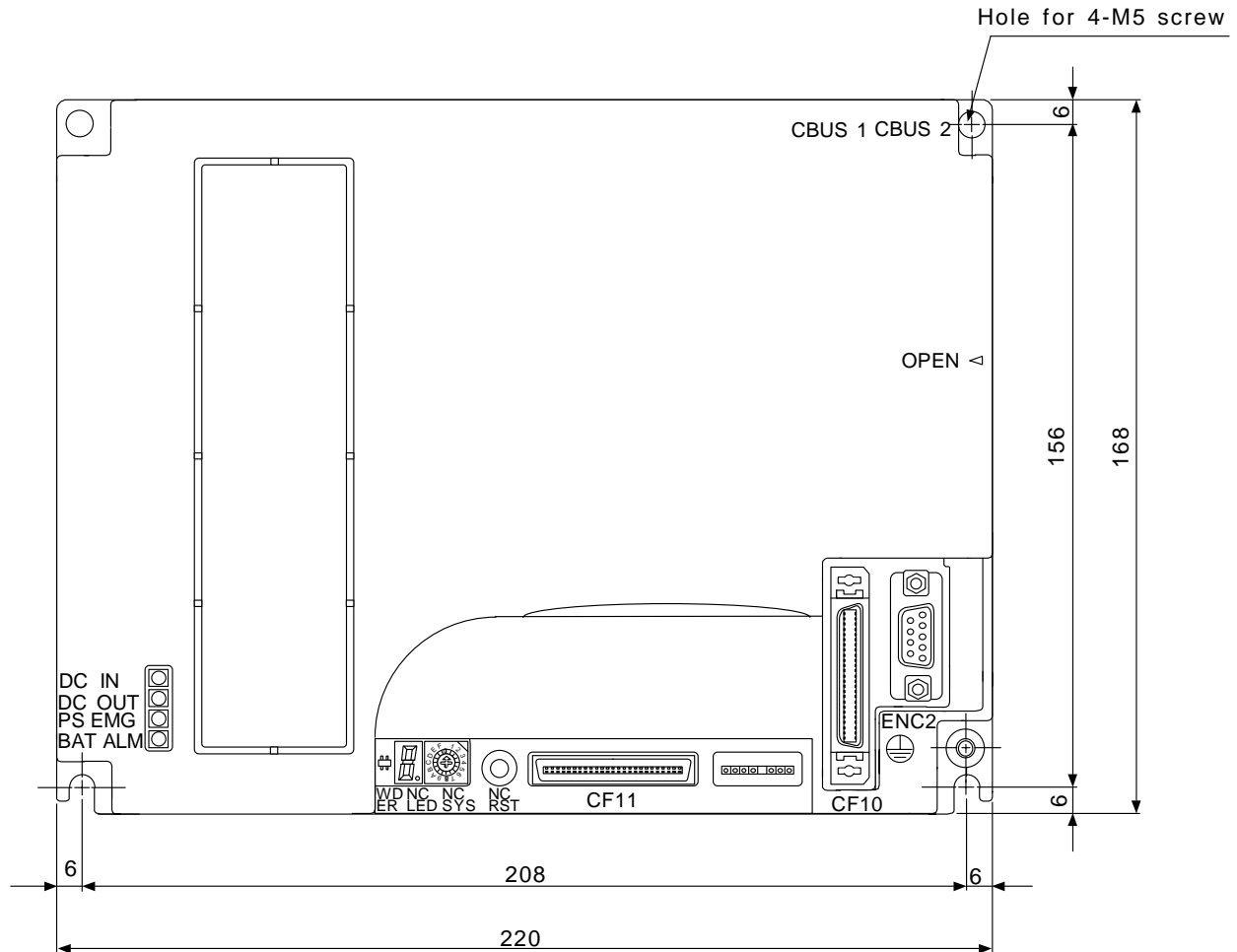
(Note 3) This denotes characters (including blanks) which are stored inside the controller and which correspond to the command codes. @ is not displayed.

Appendix 2. Outline and installation dimension drawings of units

Appendix 2. Outline and Installation Dimension Drawings of Units

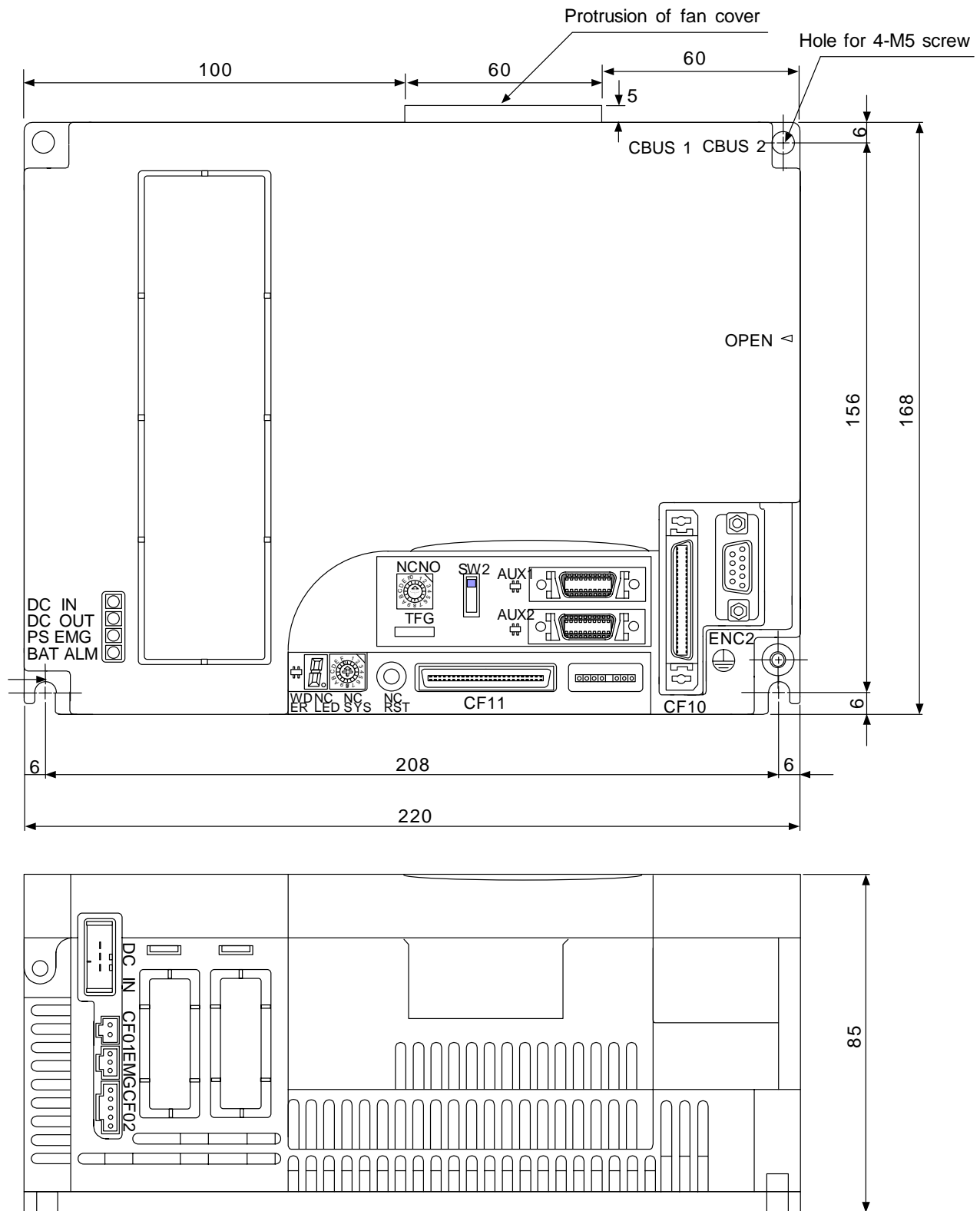
Appendix 2.1 Outline Drawing of Control Unit

Appendix 2.1.1 M64A/M64 Control Unit Outline Drawing



Appendix 2. Outline and installation dimension drawings of units

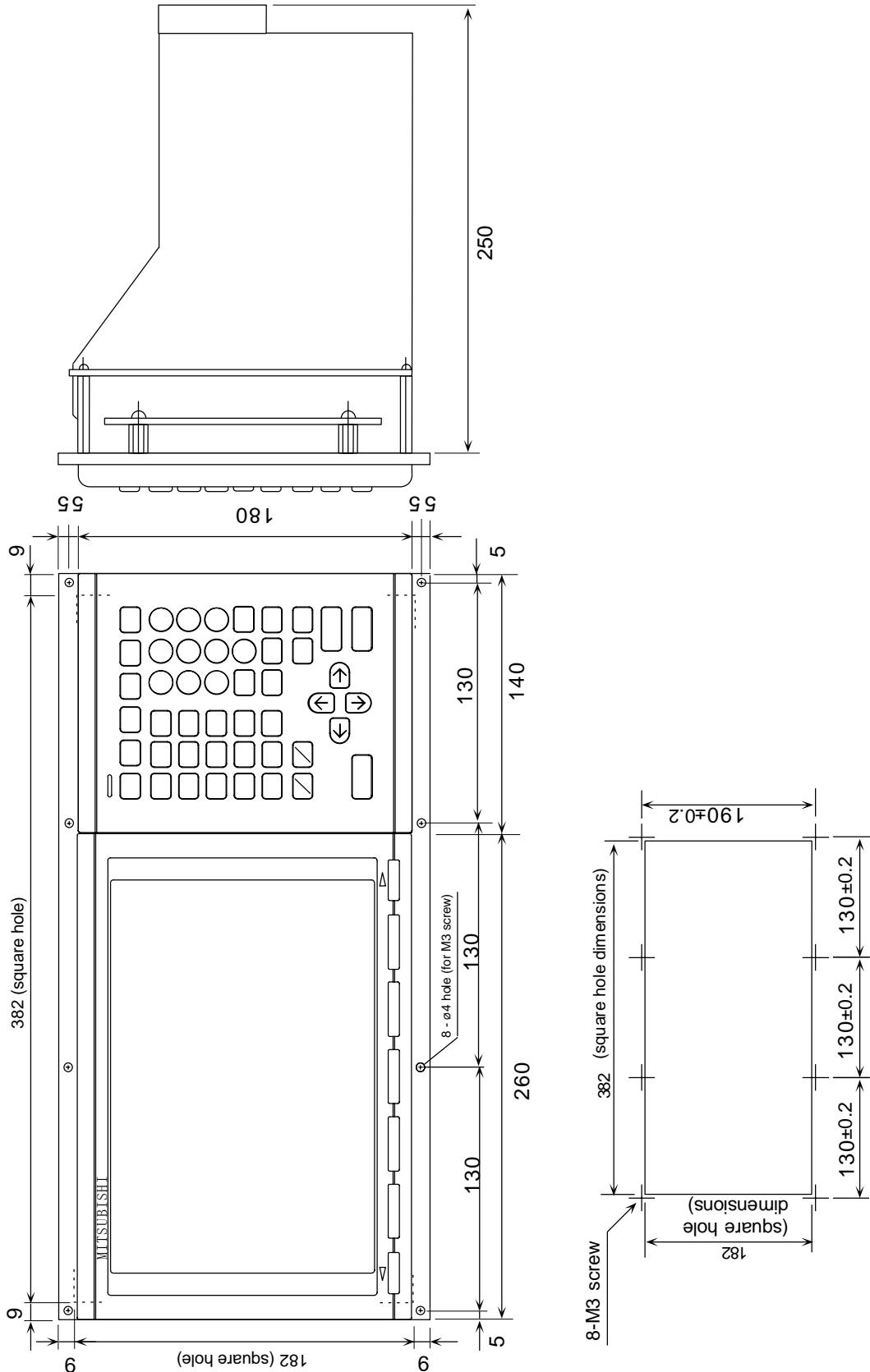
Appendix 2.1.2 M64AS/64S/65/65S/66/66S Control Unit Outline Drawing



Appendix 2. Outline and installation dimension drawings of units

Appendix 2.2 Communication Terminal Outline Drawing

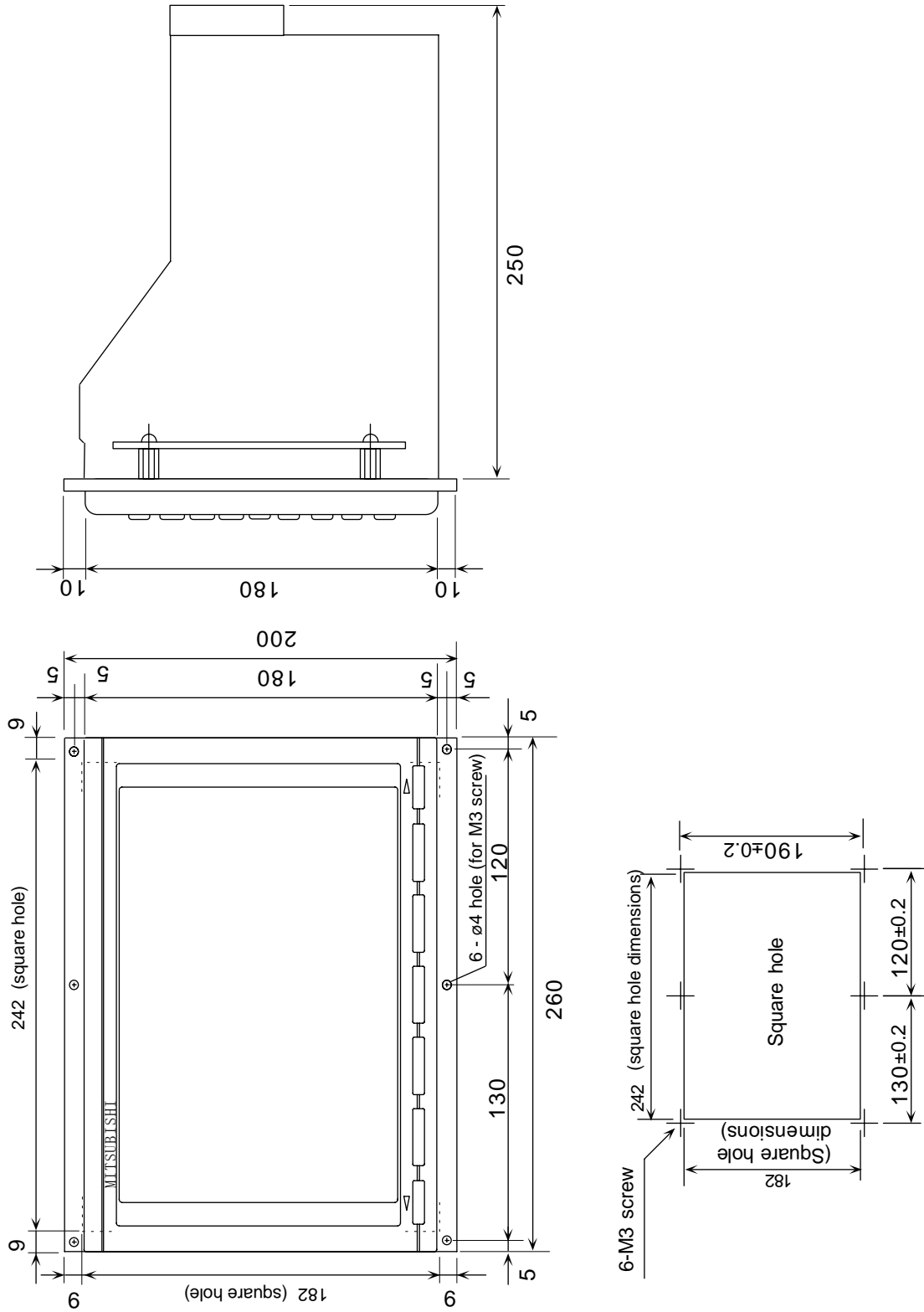
Appendix 2.2.1 FCUA-CT100/CT120



Panel cut drawing

Appendix 2. Outline and installation dimension drawings of units

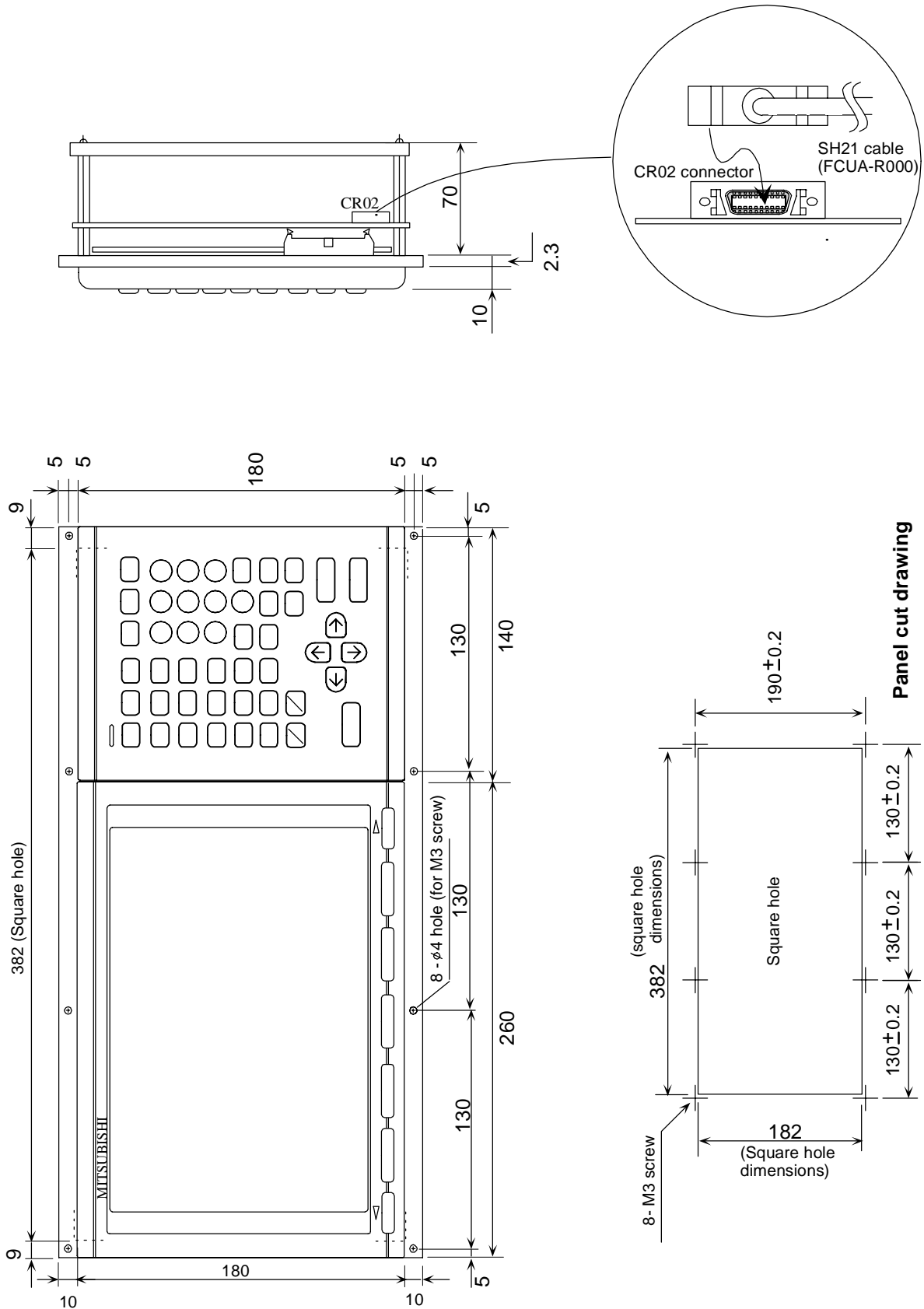
Appendix 2.2.2 FCUA-CR10



Panel cut drawing

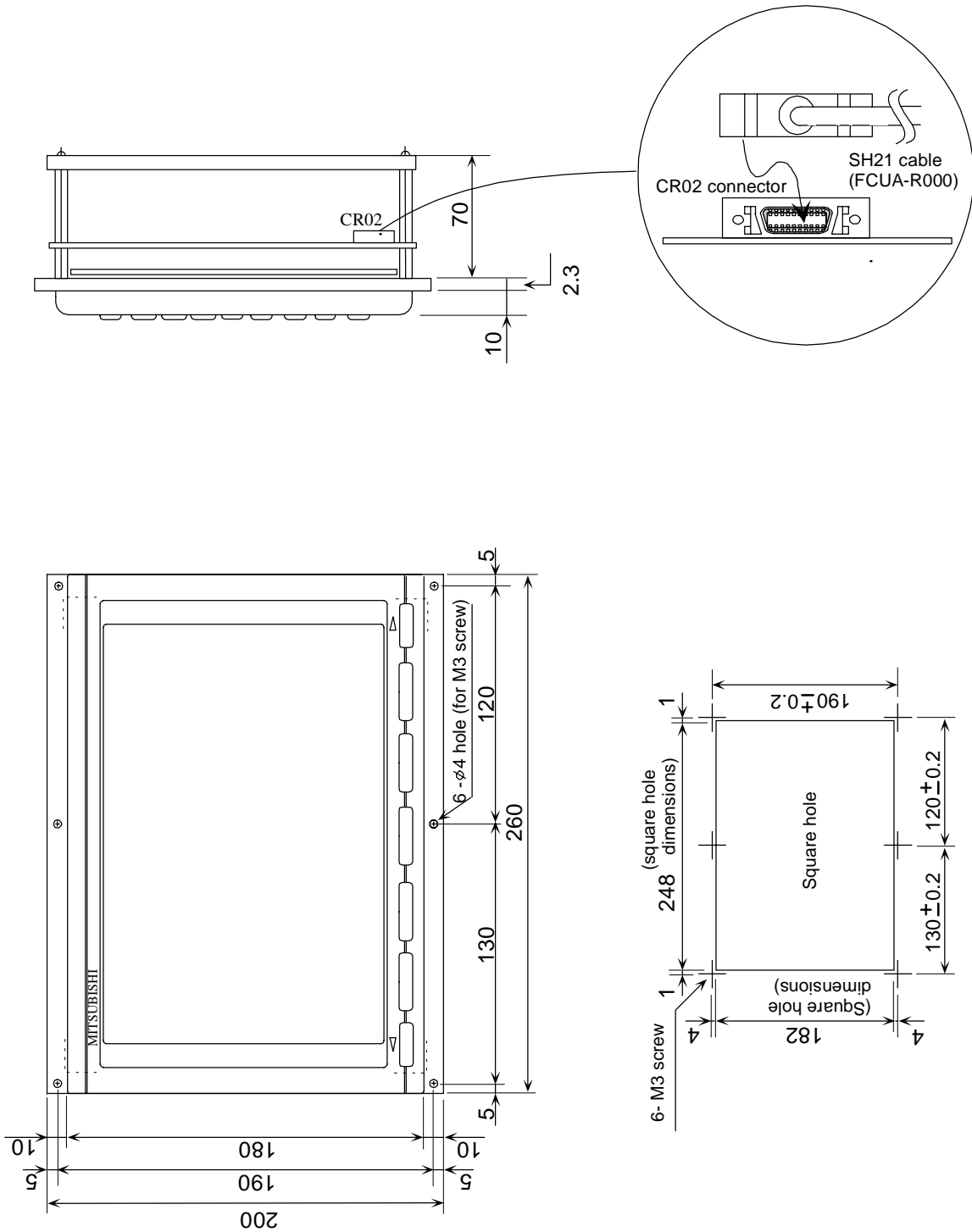
Appendix 2. Outline and installation dimension drawings of units

Appendix 2.2.3 FCUA-LD100



Appendix 2. Outline and installation dimension drawings of units

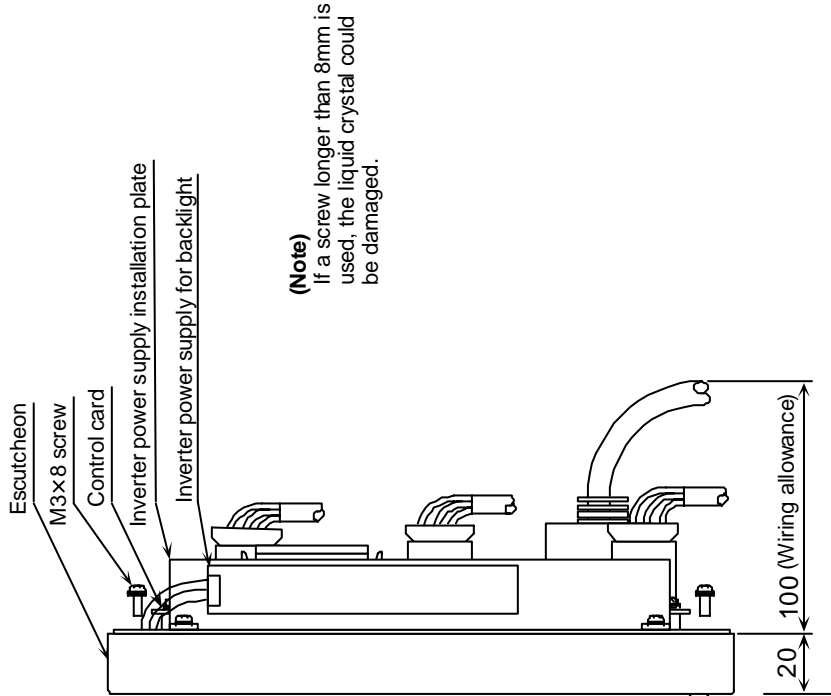
Appendix 2.2.4 FCUA-LD10



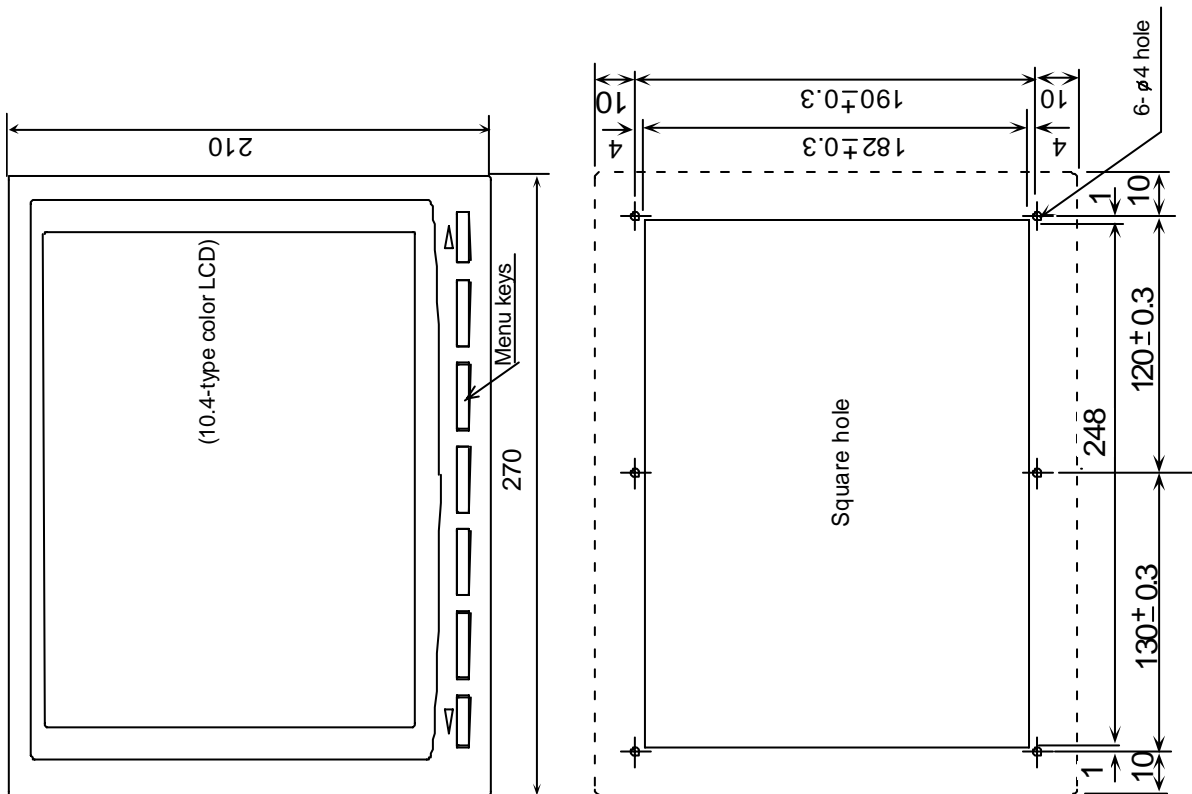
Panel cut drawing

Appendix 2. Outline and installation dimension drawings of units

Appendix 2.2.6 FCU6-DUT33

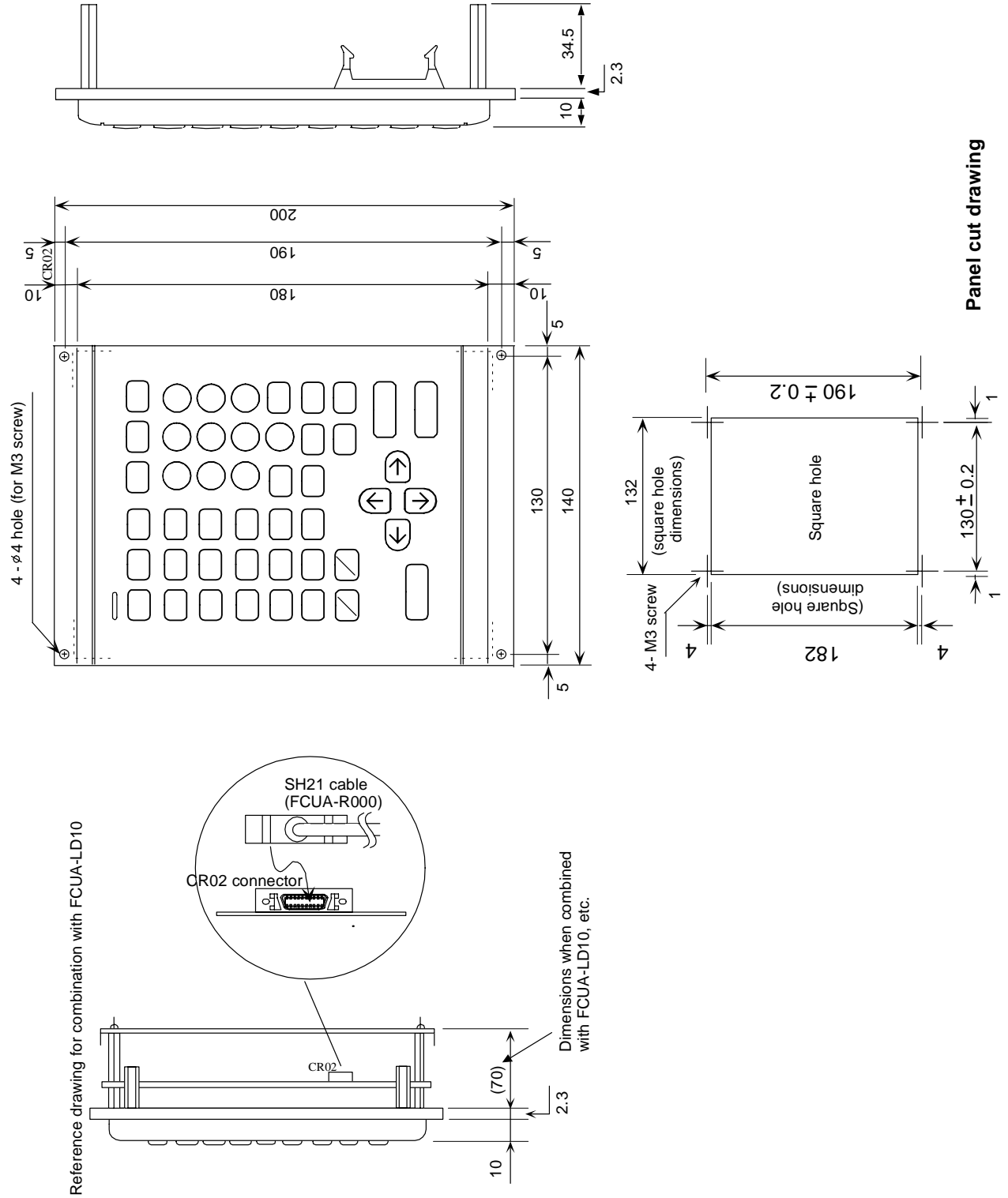


Panel cut drawing



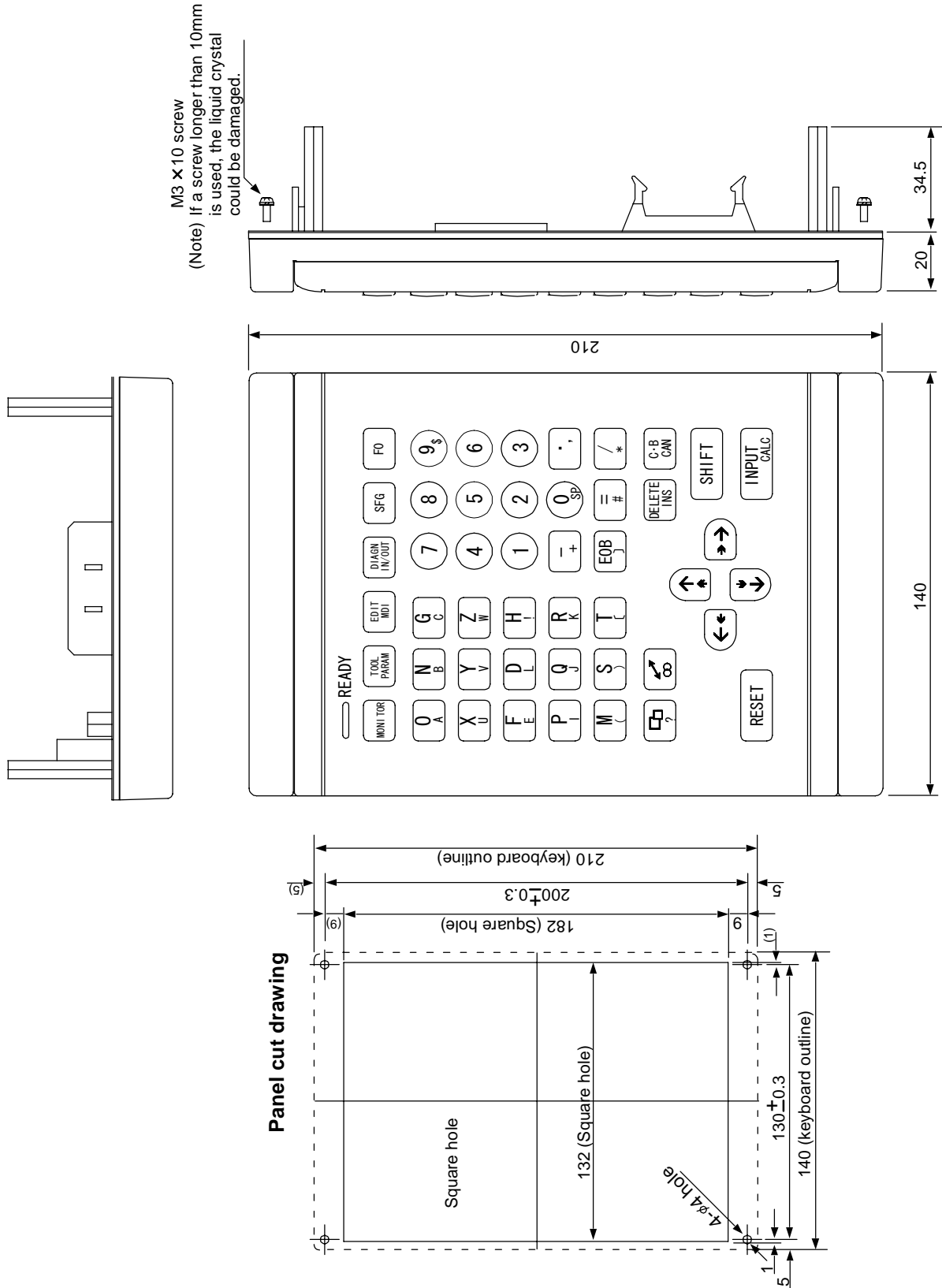
Appendix 2. Outline and installation dimension drawings of units

Appendix 2.2.7 FCUA-KB20/KB30



Appendix 2. Outline and installation dimension drawings of units

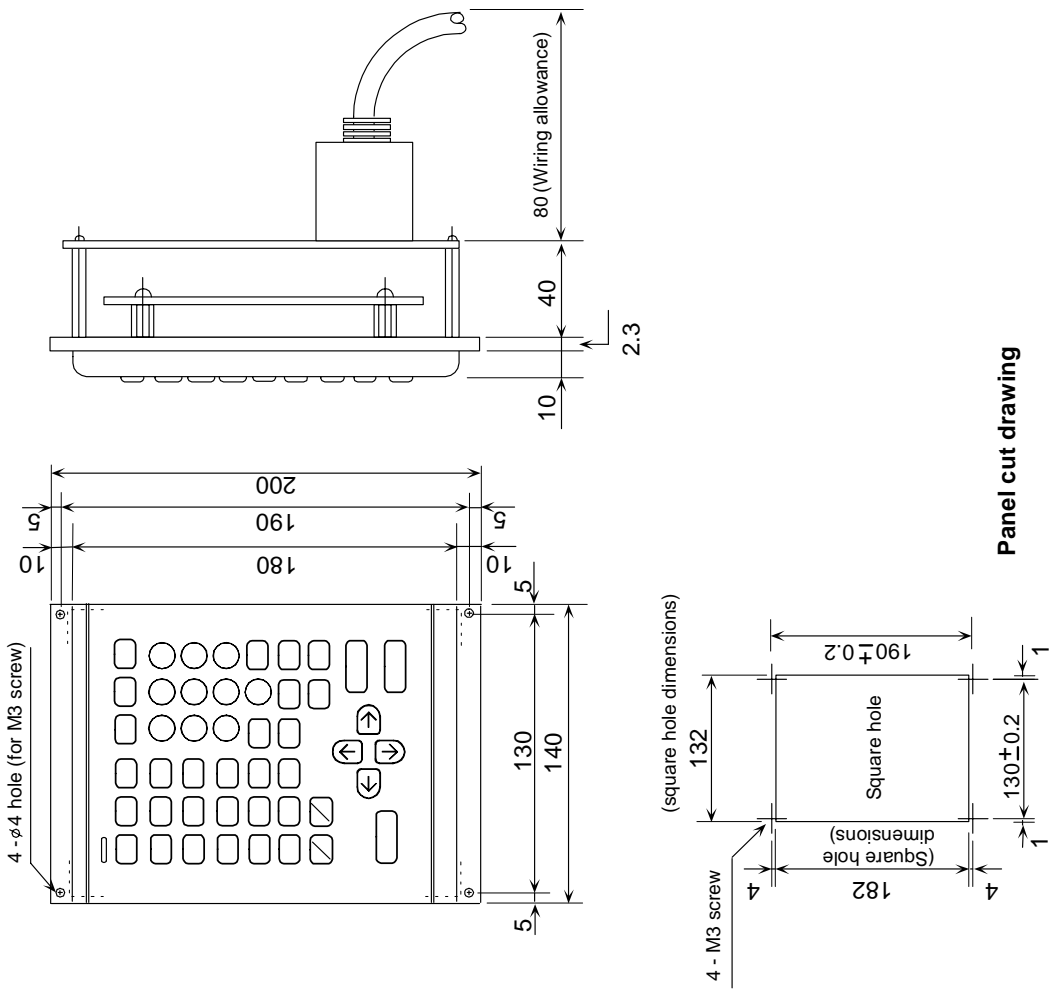
Appendix 2.2.8 FCU6-KB021/KB031



* The key layout for the FCU6-KB021 is shown in this drawing.

Appendix 2. Outline and installation dimension drawings of units

Appendix 2.2.9 FCUA-KB10/KB12

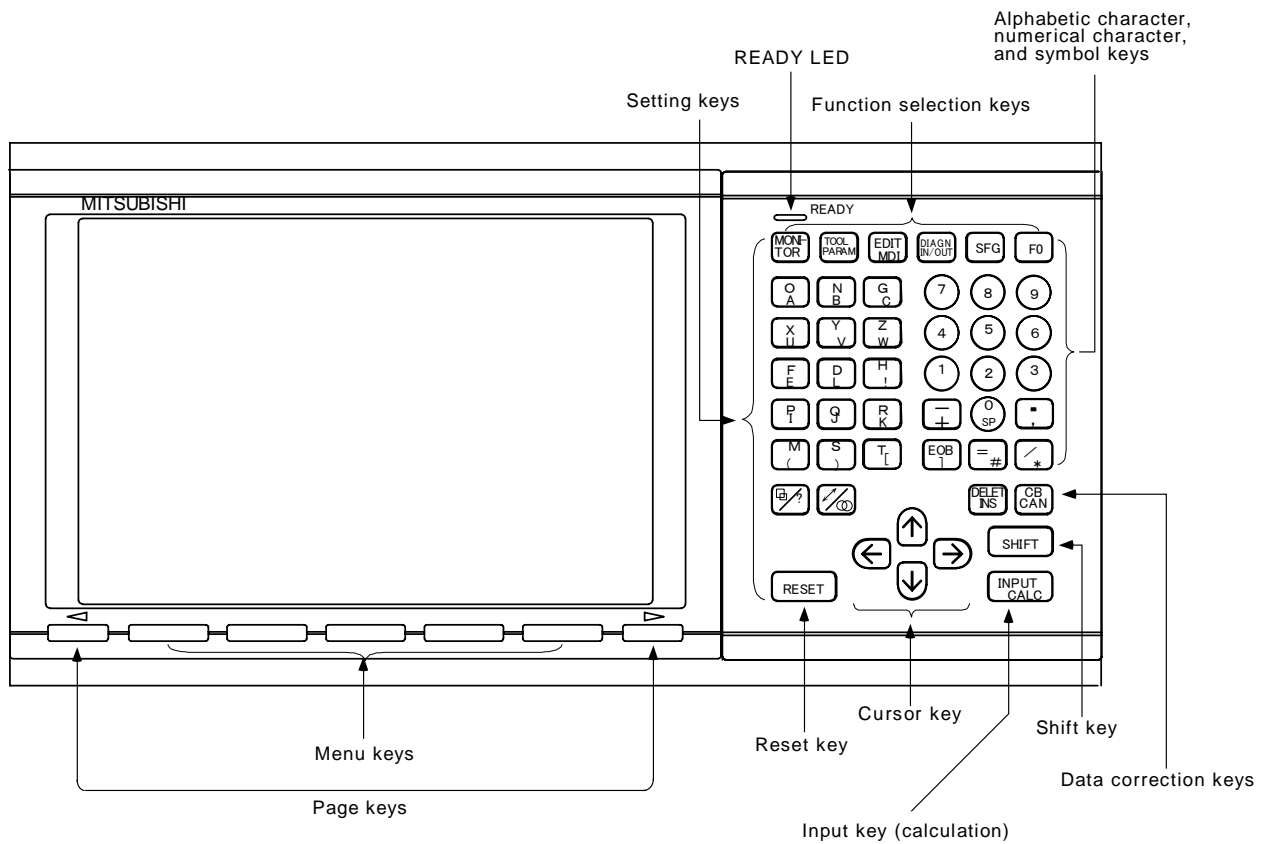


Panel cut drawing

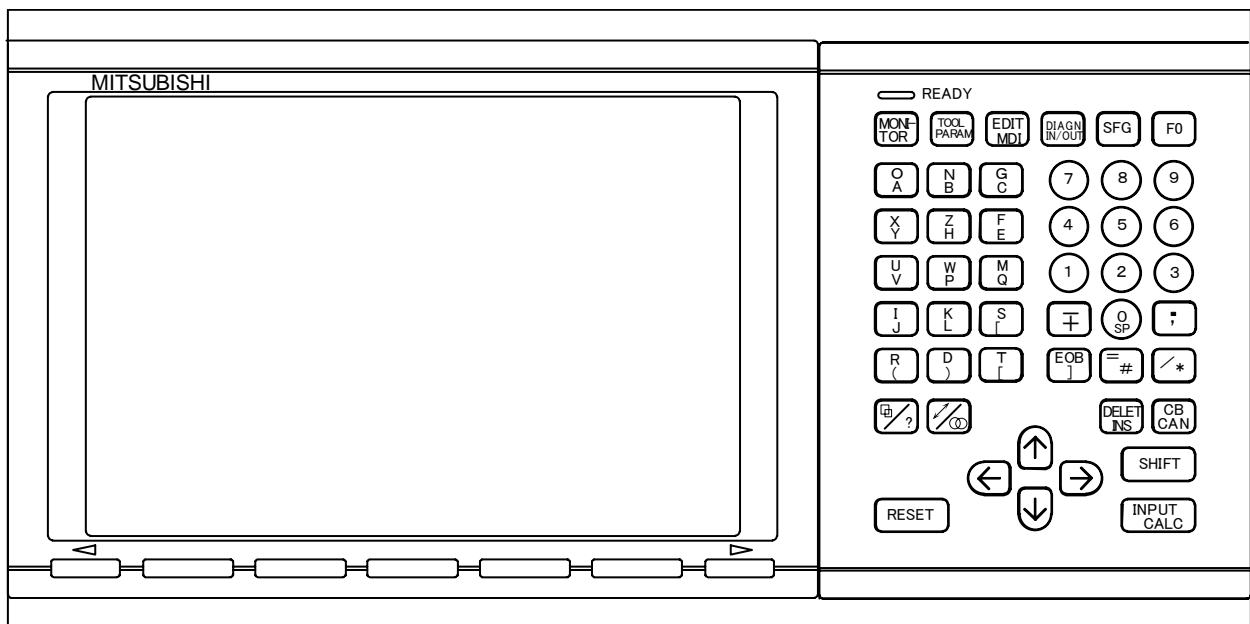
Appendix 2. Outline and installation dimension drawings of units

Appendix 2.2.10 Key Arrangement

(1) Appearance of CT100/LD100/separate type FCUA-CR10 + KB10/KB12



(2) Appearance of CT120/separate type FCUA-LD10 + FCUA-KB30

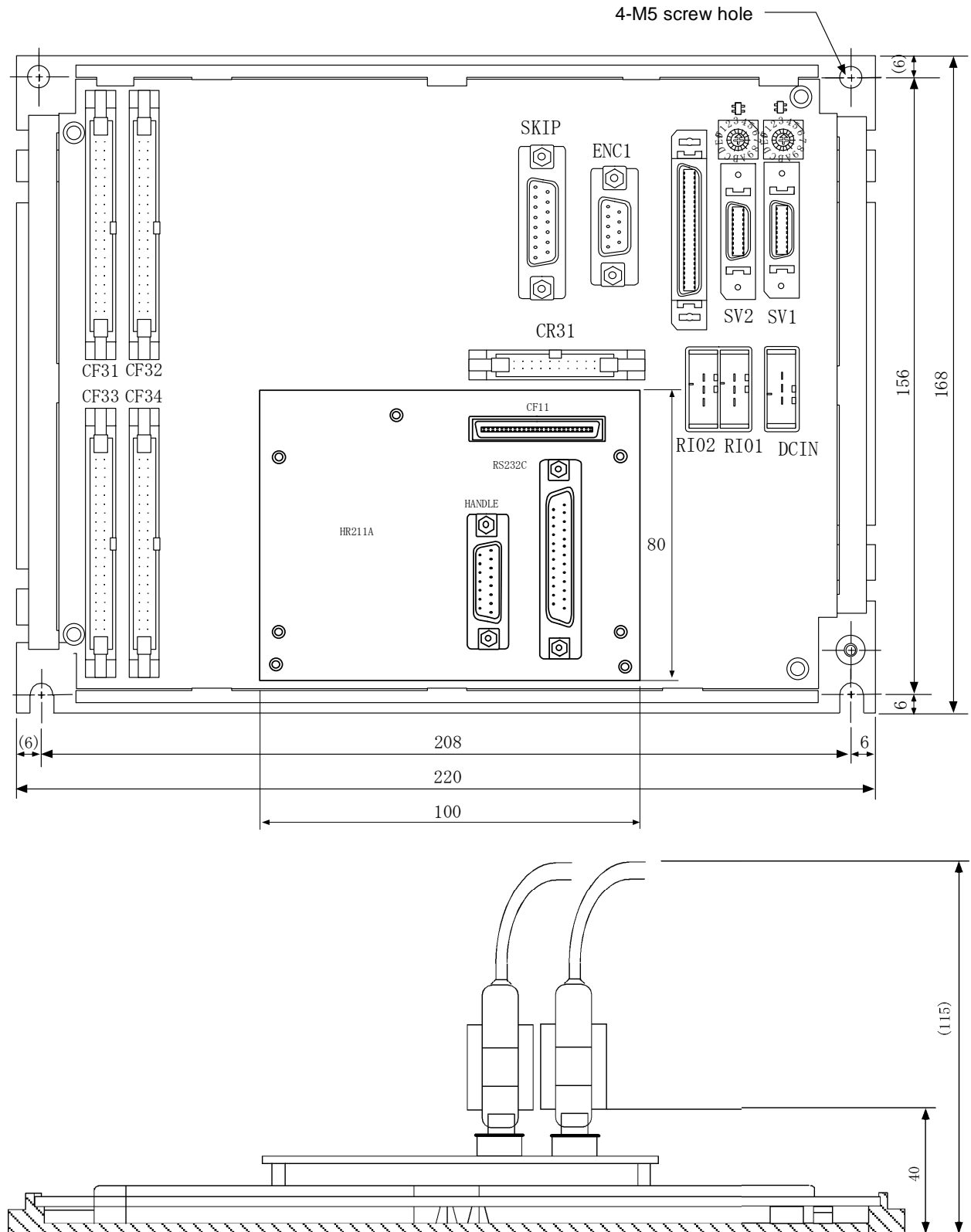


(Note) To input the alphabetic characters or symbols on the lower of the alphabetic character and symbol keys, press **SHIFT** key, then press the corresponding key.

(Example) "A" is input by pressing **SHIFT**, **O/A**.

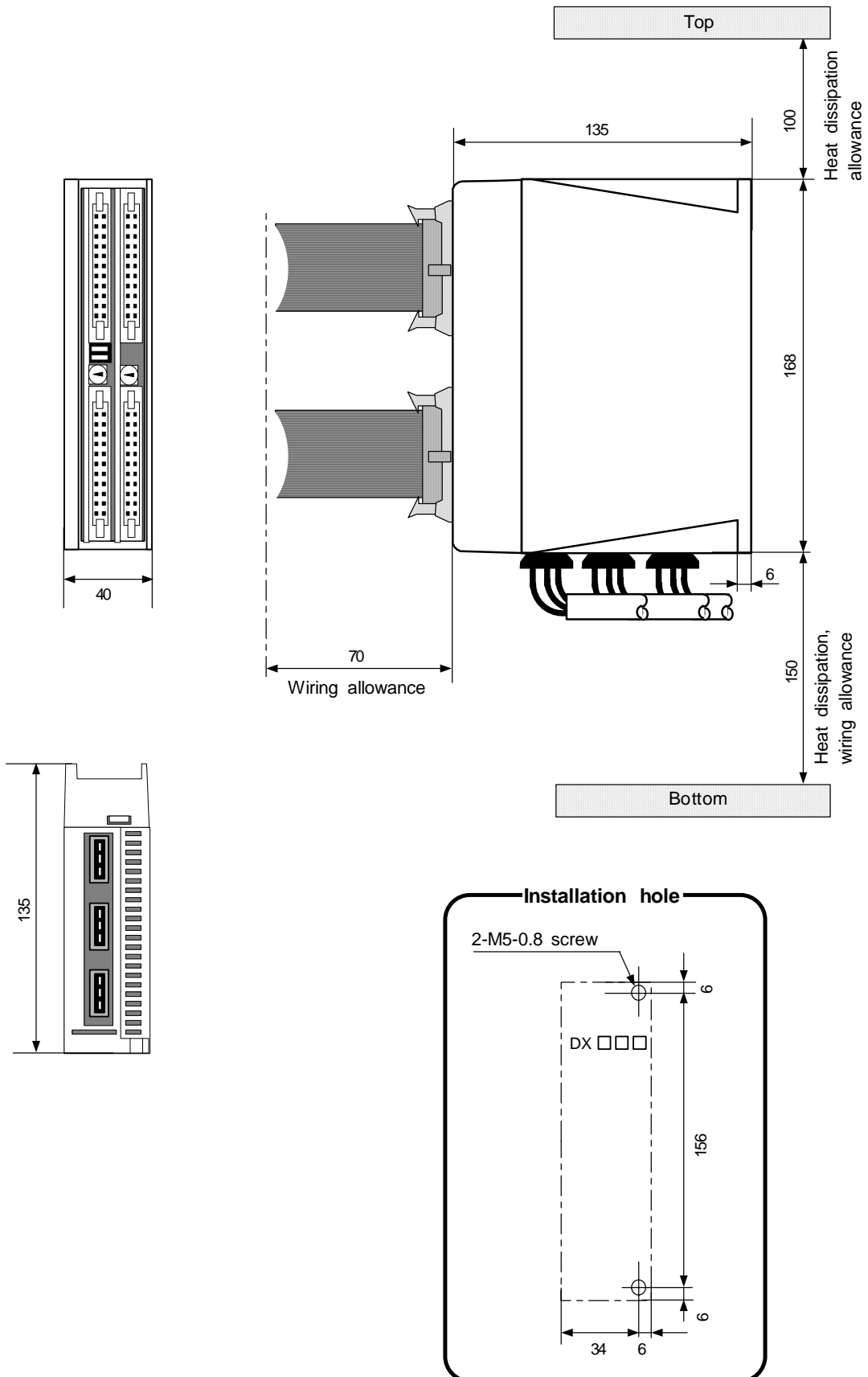
Appendix 2. Outline and installation dimension drawings of units

Appendix 2.3 Base I/O Unit Outline Drawing



Appendix 2. Outline and installation dimension drawings of units

Appendix 2.4 Remote I/O Unit Outline Drawing



Revision History

Date of revision	Manual No.	Revision details
Sep. 2001	BNP-B2210B	<ul style="list-style-type: none"> • The contents are changed according to the M64AS/M64S/M65S/M66S. • Mistakes are corrected. • Design of the cover and the back cover were changed. • MODEL, MODEL CODE, and Manual No. were added on the back cover. • The title of manual is changed to "MELDAS 60/60S Series Specifications Manual"
Apr. 2002	BNP-B2210C	<ul style="list-style-type: none"> • The contents are revised in accordance with the M60S system software version B2.
Jul. 2002	BNP-B2210D	<ul style="list-style-type: none"> • 4.2.3 Inverse Time Feed M64/65/66S Δ→ ☆ • 9.1.3 Tool Compensation for 3rd / 4th Axis M64/65/66(L) – → ○ M64/65/66S(L) – → ○ • 14.2.1.3 Manual Skip M64/65/66S Δ→ ☆ • 18.6.1 External PLC link I (M-NET) M64/65/66S ☆ → –
Mar. 2003	BNP-B2210E	<ul style="list-style-type: none"> • The contents are revised in accordance with the M60S system software version B3.
Mar. 2004	BNP-B2210F	<ul style="list-style-type: none"> • The following sections were deleted: 3.3.1, 3.3.4, 4.2.3, 4.2.5, 4.5.5, 4.5.7, 4.7.2, 6.4, 8.1.3.5, 9.3.1.4, 11.4.12, 12.1.8.4, 12.1.8.8, 14.2.1.3, 14.3.1.1, 14.3.1.2, 15.3.3, 15.3.7, Appendix 2 • The following section numbers were changed: 5.2.5 to 6 → 15.3.10 to 11, 6.3.9 to 12 → 6.3.10 to 13, 11.1.2 → 11.1.3, 12.1.2.4.8 to 9 → 12.1.2.4.9 to 10, Appendix 3 → Appendix 2 • The following sections were added: 6.3.9, 11.4.14, 12.1.2.2.1, 12.1.2.4.8, 12.1.7.1.7, 12.1.7.2.1, 12.1.8.9, 15.3.10, 15.3.11, 18.1.1.1, 18.1.12, 18.1.13, 18.2.3.1, 18.6.2.1 • The specification names and details were reviewed. • Mistakes were corrected. • M60S System software Version C0, C1 compliance

Notice

Every effort has been made to keep up with software and hardware revisions in the contents described in this manual. However, please understand that in some unavoidable cases simultaneous revision is not possible.

Please contact your Mitsubishi Electric dealer with any questions or comments regarding the use of this product.

Duplication Prohibited

This instruction manual may not be reproduced in any form, in part or in whole, without written permission from Mitsubishi Electric Corporation.

© 1998-2004 MITSUBISHI ELECTRIC CORPORATION
ALL RIGHTS RESERVED

 **mitsubishi electric corporation**
HEAD OFFICE : MITSUBISHI DENKI BLDG., 2-2-3, MARUNOUCHI, CHIYODA-KU, TOKYO 100-8310, JAPAN

MODEL	M60/60S Series
MODEL CODE	008-197
Manual No.	BNP-B2210F(ENG)