

OSP7000L

OSP700L

PROGRAMMING

(1st Edition)

Pub.No.3812-E (LE33-008-R1) July 1994

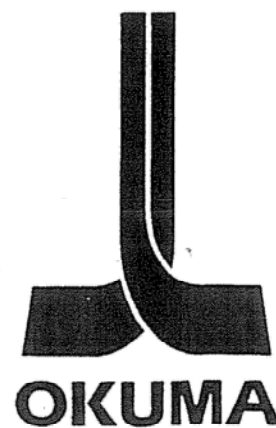


TABLE OF CONTENTS

PAGE

SAFETY PRECAUTIONS

INTRODUCTION

SECTION 1	BEFORE YOU BEGIN PROGRAMMING	1
1.	Coordinate System	3
1-1.	Models LC30/LC40/LC50	3
1-2.	Models LS30-N and LH35-N	4
1-3.	Models LH55-N	5
1-4.	LB Series Models	6
1-5.	LR Series Models	7
2.	Reference Point	8
3.	Program Zero	9
4.	Dimension Words	10
5.	Turret Selection for 2-saddle/2-turret Models	13
5-1.	LC Series	13
5-2.	LS30-N, LH35-N and LH55-N	14
5-3.	LR Series	16
6.	Tool Selection	17
6-1.	LC Series	17
6-2.	LS30-N, LH35-N and LH55-N	21
6-3.	LB Series	24
6-4.	LR Series	28
7.	Spindle Speed Selection	32
7-1.	LC30	32
7-2.	LC40	34
7-3.	LC50	36
7-4.	LS30-N	39
7-5.	LH35-N	40
7-6.	LH55-N	41
7-7.	LB9	42
7-8.	LB15-II	43
7-9.	LB25	44
7-10.	LB25-II	45
7-11.	LB35-II	46
7-12.	LB45-II	47
7-13.	LR10	48

	<u>PAGE</u>
7-14. LR15	49
7-15. LR25	50
7-16. LR35	51
7-17. LR45	52
8. Spindle Rotation Direction Selection	53
9. Tape Format	54
9-1. Program Process Sheet	54
9-2. Tape Format	55
9-3. Position of Decimal Point	56
9-4. Position of Decimal Point for 0.1 μ m Specification (optional)	58
9-5. Tape Code	59
9-6. EIA Special Codes	61
9-6-1. Setting of EIA Special Characters	62
9-6-2. Input/Output of The Special EIA Codes	62
10. Determining Cutting Conditions	64
11. Programming Codes and Word Format (1 mm (0.02 in.) Tape Input Unit System) ...	65
12. Programming Format	77
12-1. Model 1S (Single Turret Model)	77
12-2. Model 2S (Two-Turret/Two-Saddle Model)	78
SECTION 2 SIMPLE PROGRAMMING	80
1. Positioning (G00)	81
2. Straight-Line Cutting (G01)	82
3. Taper Cutting (G01)	85
4. Arc Cutting (G02/G03)	87
5. Thread Cutting (Straight, Constant Lead Thread) (G33)	91
SECTION 3 APPLICATION PROGRAM	94
1. Programming Special Threads	94
1-1. G33 (Fixed Thread Cutting Mode)	94
1-2. G32 (Fixed Thread Cutting on End Face Mode)	96
1-3. G34 and G35 (Non-Fixed Cycle)	99
2. Precautions for Programming Thread Cutting Cycles	100
3. Programming Constant Speed Cutting Operation	106
4. Incremental Programming	108

	<u>PAGE</u>
5. Supplement	109
SECTION 4 PROGRAMMING EXAMPLE	113
1. Program Example for Center-work	113
2. Program Example for Chuck Work	117
SECTION 5 PROGRAMMING TAKING TOOL NOSE RADIUS INTO CONSIDERATION	120
1. Programming Example for Taper Cutting	122
2. Programming Example for Arc Cutting	123
3. Programming Examples with Tool Nose Radius Compensation	125
4. General Programming Procedure for Circular Interpolation	126
SECTION 6 PROGRAMMING FOR SIMULTANEOUS 4-AXIS CUTS (2S Model for LC)	128
1. Programming Format	128
2. Precautions for Programming Simultaneous 4-axis Cuts	131
3. Programming Example	133
3-1. Workpiece Dimensions	133
3-2. Tooling and Cutting Conditions	133
3-3. Program Process Sheet	134
4. Waiting Synchronization M Code (M100) for Simultaneous Cuts	135
SECTION 7 BARRIER CHECK FUNCTION	136
1. General Description	136
2. Chuck Barrier and Tailstock Barrier	137
2-1. Establishing Chuck Barrier	137
2-2. Establishing Tailstock Barrier	138
2-3. Tool Movements and Alarm	139
2-4. Precautions	139
3. Tool Interference Check	140
3-1. Programming	140
3-2. Setting of Tool Interference Area	141
SECTION 8 DROOP CONTROL	146
SECTION 9 MM/MIN. MODE FEEDRATE COMMAND	147

	<u>PAGE</u>
SECTION 10 ZERO SET BY G CODE	148
SECTION 11 TAPER ANGLE/ARC RADIUS DIRECT COMMAND	149
1. General Description	149
2. Direct Taper Angle Command	149
3. Direct Radius Command	151
SECTION 12 AUTOMATIC CHAMFERING	153
1. General Description	153
2. 45° Chamfering (G75)	153
3. Rounding (G76)	155
SECTION 13 COMPOUND FIXED CYCLE (SPECIAL FIXED CYCLE)	157
1. General Description	157
2. Thread Cutting Compound Cycle (G71/G72)	158
2-1. Longitudinal Thread Cutting Cycle (G71)	158
2-2. Transverse Thread Cutting Compound Fixed Cycle (G72)	159
2-3. M Code Specifying Thread Cutting Mode and Infeed Pattern	161
2-4. P Codes in the G71 and G72 Thread Cutting Cycles (2-Saddle Specification)	172
2-5. Multi-thread Thread Cutting Function in Compound Fixed Thread Cutting Cycle	173
3. Grooving/Drilling Compound Fixed Cycle (G73/G74)	174
3-1. Longitudinal Grooving Fixed Cycle (G73)	174
3-2. Transverse Grooving Fixed Cycle (G74)	176
3-3. Axis Movements in Grooving Compound Fixed Cycle	178
3-4. P Codes in the G73 and G74 Grooving Cycles (2-Saddle Specification)	179
4. Tapping Compound Fixed Cycle (G77, G78)	180
4-1. Right-hand Tapping Cycle (G77)	180
4-2. Left-hand Tapping Cycle (G78)	181
4-3. Supplement	182
5. Application of Compound Fixed Cycle	183
5-1. Application of Longitudinal Thread Cutting Compound Fixed Cycle (G71) ...	183
5-2. Application of Longitudinal Grooving Compound Fixed Cycle (G73)	184
5-3. Application of Transverse Grooving Compound Fixed Cycle (G74)	184

	<u>PAGE</u>
SECTION 14 MIRROR IMAGE FUNCTION (2-Turret Model)	185
1. Outline of Mirror Image Functions	185
2. Operations	186
3. Program Axis Motions	187
3-1. G Codes	187
3-2. Cautions on Programming	188
3-3. Cutting Program	189
3-4. Cutting Operation	190
4. Machine Operation and Display	191
4-1. Operation Panel	191
4-2. Zero Offset	191
4-3. Tool Offset	192
5. Others	193
6. Program Example	194
SECTION 15 USER TASK 1	196
1. Outline of User Task Function	196
2. Fundamental Functions of User Task	197
2-1. Control Statement	198
2-1-1. GOTO Statement - Unconditional Jump	199
2-1-2. IF Statement - Conditional Jump	199
2-2. Common Variable	201
2-2-1. Common Variable	201
2-2-2. Local Variables	202
2-2-3. System Variables	203
2-3. Arithmetic Operation Function	210
2-3-1. Arithmetic Expression	210
2-3-2. Comparison Expression	210
3. Program Example	211
3-1. PSELECT Command	213
3-2. VSET Command	214
3-3. End Block of Schedule Program	214
4. Comparison of User Task 1 and User Task 2	215

	<u>PAGE</u>
SECTION 16 TOOL NOSE RADIUS COMPENSATION FUNCTION	216
1. Outline	216
1-1. General Description	216
2. Tool Nose Radius Compensation for Turning Operation	216
2-1. Tool Offset and Nose Radius Compensation	216
2-2. Tool Nose Radius Compensation at Discontinuous Point	217
3. Compensation Operation	218
3-1. Geometrical Cutting Error Due to Tool Nose Radius	218
3-2. Compensation Movement	219
4. Programming	220
4-1. G Codes	220
4-2. T Codes	221
4-3. Program Example	222
5. Tool Nose Radius Compensation Data	223
5-1. Measuring Nose Radius	223
5-2. Setting Compensation Amounts	225
5-3. Relationship between Imaginary Tool Tip Orientations and Coded Numbers ..	228
6. Display	230
6-1. Actual Position	230
6-2. Alarm Display	230
7. Operation	231
7-1. Conditions of Nose Radius Compensation Function	231
7-2. Path of Tool Nose "R" Center in Tool Nose Radius Compensation Mode ...	233
8. Application of Tool Nose Radius Compensation Programming	235
8-1. G41 and G42	235
8-2. Behavior on Tool Nose Radius Compensation Mode Entry	236
8-2-1. General Description	236
8-2-2. Program Examples	237
8-2-3. I and K Command with G41 and G42	240

	<u>PAGE</u>
8-3. Behavior in Tool Nose Radius Compensation Mode	241
8-3-1. Straight Line to Straight Line Cutting	241
8-3-2. Straight Line to Arc Cutting (Arc to Straight Line Cutting)	251
8-3-3. Arc to Arc Cutting	255
8-3-4. Switching from G41 to G42 or from G42 to G41	256
8-4. Behavior with Tool Nose Radius Compensation Mode Cancel	259
8-4-1. G40 Given with X- or Z-axis Motion Command	259
8-4-2. I and K Command with G40	262
8-4-3. Independent G40	263
8-5. Relieving Tool to Change 'S' or 'M' Code during Cutting	264
8-5-1. Program 2	265
8-5-2. Program 3	266
8-5-3. Program 4	267
8-5-4. Program 5	268
8-5-5. Program 6	269
8-6. Precautions	270
SECTION 17 AUTOMATIC ANY-ANGLE CHAMFERING	271
1. Overview of Automatic Any-Angle Chamfering Function	271
2. Programming Examples	271
2-1. C-Chamfering (G75)	271
2-2. R-Chamfering (G76)	272
3. Precautions	273
SECTION 18 TURRET UNCLAMP COMMAND (for NC Turret Specification)	274
APPENDIX 1 G CODE TABLE	275
APPENDIX 2 M CODE TABLE	281
APPENDIX 3 SYSTEM VARIABLES LIST	289