PROGRAMMING

MAR. 1989
EDITION 0-1

CONTENTS

1. THE OUTLINE OF THE PROGRAMMING 1
1-1 Regular Procedure for Making Program 1
1-2 The Motion of the Turret Slide 2
1-3 The Cutting Area 2
1-4 Part Dimensioning 6
1-5 Part Layout Sheet 7
1-6 Process Sheet 9

2. TAPE INFORMATION 10
2-1 Move Command for the Cutting Tools 12
2-2 The Absolute and the Incremental Input 13
2-3 The Program Format 15
2-4 G-Function (Preparatory Function) 19
[1] G50 (Maximum spindle speed setting) 19
[2] G00 (Positioning) 20
[3] G01 (Linear interpolation, 45° Chamfering, Rounding) 21
[4] G02, G03 (Circular interpolation, Radius programming for circular interpolation) 27
[5] G04 (Dwell) 32
[6] G10 (Changing of tool offset amount programmable data input) 34
[7] G20,G21 (Inch/metric input) 35
[8] G25, G26 (Spindle speed fluctuation detecting function) 36
[9] G27 (Reference point return check) 37
[10] G28 (Automatic return to reference point) 38
[11] G30 (Second reference point return) 39
[12] G31 (Skip function) 40
[13] G32 (Thread cutting) 41
[14] Other G-codes 54
[15] List of G-codes and their group 56
2-5 F-Function (Feed Function) 58
[1] Feed rate per revolution of the spindle (G99) 58
[2] Feed rate per minute (G98) 59
2-6 S-Function (Spindle Speed Function) 61
[1] Spindle RPM direct command (G97 mode) 61
[2] Cutting speed command (G96 mode) 63
2-7 Coordinate System Setting 64
2-8 T-Function (Tool Function) 67
[1] Designation of tool selection/geometry offset (tool shape) number 67
[2] Tool offset (General) 68
[3] Tool position offset (Wear offset) 69
[4] Notes of T-function 73
[5] Tool nose radius compensation 73
2-9 M-Function (Miscellaneous Function) 74
2-10 Other Function 86
[1] '/' Block delete (Optional Block Skip) 86

3. EXAMPLE OF PROGRAMMING OF VARIOUS OPERATIONS 87
3-1 Facing Operation 87
3-2 Turning Operation 89
3-3 Drilling Operation 90
3-4 Boring Operation 91
3-5 Grooving Operation 92

LE-11/21 PRO OTB 89/3

3-6 Taper Cutting and Chamfering Operation 93
3-7 Radius Cutting Operation 94
3-8 Cutting off Operation 96
3-9 Bar Pull out Operation 98
3-10 Threading Operation 101

4. TOOL NOSE RADIUS COMPENSATION (G40 to G42) 103
4-1 Different Dimension by Tool Nose Radius 103
4-2 The Motion of Tool Nose Radius Compensation 105
4-3 Command of Tool Nose Radius Compensation 106
4-4 Prohibited command on the compensation 110

5. EXAMPLE OF PROGRAMMING 114
5-1 Part Dimensioning 114
5-2 Part layout Sheet 115
5-3 Work Coordinate System Setting and Tool Selecting Position 116
5-4 Process Sheet 120

6. CANNED CYCLE 124
6-1 G90 (Turning Cycle) 124
6-2 G92 (Thread Cutting Cycle) 129
6-3 G94 (Facing Cycle) 135

7. MULTIPLE REPETITIVE CUTTING CYCLE 139
7-1 G71 (Stock Removal Cutting Cycle in Turning) 139
7-2 G72 (Stock Removal Cutting Cycle in Facing) 144
7-3 G73 (Pattern Repeated Cutting Cycle) 149
7-4 G70 (Finish Cutting Cycle) 153
7-5 G74 (Longitudinal Pecking Cycle) 154
7-6 G75 (Cross Pecking Cycle) 157
7-7 G76 (Thread Cutting Cycle) 160
7-8 Notes on Multiple Repetitive Cycles (G70 to G76) 166

8. DRAWING DIMENSION DIRECT INPUT (OPTION) 167
[1] Instruction method 168
[2] Commanding of angle, arc and chamfer 169
[3] Commands table 170

9. STANDARD PROGRAM STYLE 174

10. SAMPLE PROGRAMMING 179
CHUCK WORK 180
SHAFT WORK 185
BAR WORK 192

11. LIST OF TAPE FORMAT FOR LE-11 AND LE-21 199
11-1 Preparatory Function (G Function) 199
11-2 Miscellaneous Function (M Function) 201
11-3 Tool Function (T Function) 202
11-4 Spindle Function (S Function) 202
11-5 Multiple Repetitive Cycle 203
11-6 Canned Cycle 206