

PROGRAMMING MANUAL

MORI-FANUC 0T

MFNC-L3(F-10T)

MORI-FANUC 11T

MORI-FANUC 15T



MORI SEIKI

MORI SEIKI CO., LTD.

NARA JAPAN

CONTENTS

SPECIFICATION

1. MORI FANUC 11T Specification	P-1
2. MFNC-L3 (FANUC-10T) Specifications	P-4
3. MORI FANUC 0T Specifications	P-7
 I. First Section	P-10
I-1 Programming terms	P-11
I-2 G Functions	P-17
I-2-(1) G00 (Rapid feed)	P-18
G01 (Linear cutting)	P-18
I-2-(2) G50 Maximum spindle speed setting	P-20
I-2-(3) G96 Constant surface speed control (Cutting speed command) G97 Spindle speed command	P-21
I-3 M Function	P-23
I-3-(1) M01 Optional stop	P-24
I-3-(2) M03 Spindle normal rotation M05 Spindle stop	P-25
I-3-(3) M08 Cutting oil discharge M09 Cutting oil stop	P-26
I-3-(4) M30 End of program	P-27
I-3-(5) M41 Spindle low-speed range M42 Spindle high-speed range	P-28
I-4 S Function (spindle function)	P-29
I-5 T Function (Tool function)	P-31
I-6 F Function (Feed function)	P-32
I-7 Basic programming pattern (chuck work)	P-33
I-8 Programming procedure	P-37
I-9 Tool nose radius compensation	P-53
I-10 Exercise 1	P-60
Exercise 2	P-71
 II Second Section	P-87
II-1 G Function	P-88
II-1-(1) G02/G03 Circular cutting	P-89
II-1-(2) G04 Dwell	P-94
II-1-(3) G70 Finishing cycle G71 O.D. (I.D.) rough cutting cycle (Type1)	P-96
II-1-(4) G74 Peck drilling cycle	P-105
II-1-(5)-A G76 Multiple thread cutting cycle [for F10T, F11T, F15T]	P-107
II-1-(5)-B G76 Multiple Thread Cutting Cycle [for F0T]	P-110
II-1-(6) G92 Thread cutting cycle	P-117
II-2 M Functions	P-120
II-2-(1) M00 Temporary program stop	P-121
II-2-(2) M03, M04, M05 Spindle Start and Stop	P-122
II-2-(3) M23, M24 Chamfering	P-123
II-3 Summary of the command method [F10T, F11T, F15T]	P-125
Summary of the command method [F0T]	P-126
 III Appendix	P-127
1. Before Programming	
1) Command Points for Cutting Tools	P-128
2) Decimal Point Input	P-129
2. G Function	
G Code List	P-132
1) G00 Positioning	P-135
2) G01 Linear Interpolation	P-136

2.1) Chamfering	P-136
3) G04 Dwell	P-142
4) G27 Zero Return Check	P-143
5) G28 Automatic Zero Return	P-144
6) G29 Automatic Zero Return from Original Point	P-146
7) Thread Cutting	P-147
7.1) Precautions for Thread Cutting	P-147
7.2) Feed (Lead) Command for Thread Cutting	P-148
7.3) Example of thread Cutting	P-150
8) G50 Absolute Coordinates Preset	P-155
8.1) Automatic Coordinate System Setting	P-159
9) G98 Feed Rate Per Time (mm/min, in/min)	
G99 Feed Rate Per Spindle Rotation (mm/rev, in/rev)	P-162
3. M FUNCTION	
M Code List	P-163
1) M00 Program Temporary Stop	P-165
2) M01 Optional Stop	P-167
3) M02 Program End	P-168
4) M03, M04, M05 Spindle Start and Stop	P-170
5) M10, M11, Chuck Clamping and Unclamping	P-171
6) M25, M26 Tail Stock Spindle Out, In	P-173
7) M17, M18 Tool Post Rotation Normal, Reverse	P-174
8) M21, M22 Tailstock Forward, Backward	P-175
9) M41, M42 Spindle Speed Range Selection	P-176
10) M73, M74 Parts Catcher Out, In	P-177
11) M98, M99 Sub Program Control	P-178
11.1) Command System	P-178
11.2) Precautions in Programming	P-179
11.3) Memory Capacity	P-181
11.4) Programming Example [F10T, F11T, F15T]	P-181
4. S FUNCTION	
Spindle Function (S Function)	P-182
1) Command system	P-182
2) Precautions in programming	P-182
5. F FUNCTION	
Feed Function (F Function)	P-185
1) Command system	P-185
2) Feed per spindle rotation	P-185
2.1) Command	P-185
2.2) Cutting feed range	P-187
2.3) Cutting feed limit	P-187
3) Feed per time	P-187
3.1) Command	P-187
3.2) Feed rate range	P-188
4) Precaution on the use of F function	P-190
5) Theoretical finish surface	P-191
6. T FUNCTION	
Tool Function (T Function)	P-192
1) Command System	P-192
2) Typical Example	P-192
3) Tool Offset	P-193
4) Special Application Example of Tool Offset	P-194
4.1) Adjustment of Dimensions of Two More Places	P-194
4.2) Taper Angle Adjustment	P-196
4.3) Grooving Width Adjustment	P-197

7. CUTTING TOOL NOSE RADIUS COMPENSATION BY MANUAL CALCULATION

1) Taper Cutting	P-199
1.1) Programming examples	P-199
1.2) Offset amount table	P-204
2) Circular cutting	P-210
2.1) How to obtain command point	P-210
2.2) Programming example	P-211
3) Triangles	P-214
3.1) Ratios of Sides of Frequently Used Triangles	P-214
3.2) Definition of Trigonometric Functions	P-214
3.3) Characteristics of Triangles	P-215

SPECIAL FUNCTION

1. MULTIPLE REPETITIVE CYCLES

1) G72 Rough Cutting Cycle (Type I)	P-216
1.1) Movement	P-216
1.2) Command Method	P-216
1.3) Signs for U and W	P-217
1.4) Precautions	P-217
2) G72 Face Rough Cutting Cycle (Type II)	P-217
2.1) Movement	P-217
3) G73 Closed Loop Cutting Cycle	P-218
3.1) Movement	P-218
3.2) Command Method	P-218
3.3) Precautions	P-219
4) G74 Peck Drilling Cycle	P-220
4.1) Movement	P-220
4.2) Command method	P-220
5) O.D. Grooving Cycle	P-222
5.1) Movement	P-222
5.2) Command Method	P-222
6) Programming Examples	
6.1) G71 Rough Cutting Cycle (O.D.)	
G70 Finish Cutting Cycle	P-223
6.2) G71 Rough Cutting Cycle (I.D.)	
G70 Finish Cutting Cycle	P-224
6.3) G72 Face Rough Cutting Cycle	
G70 Finish Cutting Cycle	P-225
6.4) G73 Closed Loop Cutting Cycle	
G70 Finish Cutting Cycle	P-226
6.5) G74 O.D. Cutting and Drilling	P-227
6.6) G75 Chamfering and Stick Work	P-228
6.7) G76 Threading Cutting Cycle (P1 type)	P-229

2. AUTOMATIC TOOL NOSE RADIUS COMPENSATION PROCEDURE

Introduction	P-230
1) Tool Nose Radius	P-231
2) Requirements for Use of Nose R Compensation Function	P-232
2.1) Command of Command Point (Imaginary Tool Nose) Position	P-233
2.2) Command of Tool Nose R	P-235
3) Tool Nose R Compensation Function	P-236
3.1) Special Terms Used for Tool Nose R Compensation Function	P-237
3.1.1) Start-up	P-237
3.1.2) Offset Mode	P-237
3.1.3) Cancel Mode	P-237
3.2) Basic Compensation Movement	P-238
3.2.1) Start-up	P-238

3.2.2) Offset Mode	P-239
3.2.3) Offset Cancel	P-240
3.2.4) When Work Wall is present at the End Point of Cutting	P-241
3.3) Precautions for Tool Nose R Compensation	P-243
4) Program Examples	P-247
Appendix	P-256