



GE Fanuc Automation

Computer Numerical Control Products

Series 15i / 150i – MA

Operator's Manual (Programming)

GFZ-63324EN/02

January 2000

TABLE OF CONTENTS

SAFETY PRECAUTIONS	s-1
I GENERAL	1
1 GENERAL	3
1.1 GENERAL FLOW OF OPERATION OF CNC MACHINE TOOL	5
1.2 NOTES ON READING THIS MANUAL	7
II PROGRAMMING	9
1 GENERAL	11
1.1 TOOL MOVEMENT ALONG WORKPIECE PARTS FIGURE-INTERPOLATION	12
1.2 FEED-FEED FUNCTION	14
1.3 PART DRAWING AND TOOL MOVEMENT	15
1.3.1 Reference Position (Machine-Specific Position)	15
1.3.2 Coordinate System on Part Drawing and Coordinate System Specified by CNC - coordinate System	16
1.3.3 How to Indicate Command Dimensions for Moving the Tool - Absolute, Incremental Commands	19
1.4 CUTTING SPEED - SPINDLE SPEED FUNCTION	21
1.5 SELECTION OF TOOL USED FOR VARIOUS MACHINING - TOOL FUNCTION	22
1.6 COMMAND FOR MACHINE OPERATIONS - MISCELLANEOUS FUNCTION	23
1.7 PROGRAM CONFIGURATION	24
1.8 TOOL FIGURE AND TOOL MOTION BY PROGRAM	27
1.9 TOOL MOVEMENT RANGE - STROKE	28
2 CONTROLLED AXES	29
2.1 CONTROLLED AXES	30
2.2 AXIS NAME	31
2.3 INCREMENT SYSTEM	32
2.4 MAXIMUM STROKE	34
3 PREPARATORY FUNCTION (G FUNCTION)	35
4 INTERPOLATION FUNCTIONS	39
4.1 POSITIONING (G00)	40
4.2 SINGLE DIRECTION POSITIONING (G60)	42
4.3 LINEAR INTERPOLATION (G01)	44
4.4 CIRCULAR INTERPOLATION (G02,G03)	46
4.5 HELICAL INTERPOLATION (G02,G03)	51

4.6	HELICAL INTERPOLATION B (G02,G03)	53
4.7	HYPOTHETICAL AXIS INTERPOLATION (G07)	54
4.8	POLAR COORDINATE INTERPOLATION (G12.1,G13.1)	57
4.9	CYLINDRICAL INTERPOLATION (G07.1)	63
4.10	CYLINDRICAL INTERPOLATION CUTTING POINT CONTROL (G07.1)	67
4.11	EXPONENTIAL INTERPOLATION (G02.3,G03.3)	81
4.12	INVOLUTE INTERPOLATION (G02.2,G03.2)	89
4.12.1	Involute Interpolation with a Linear Axis and Rotation Axis (G02.2,G03.3)	95
4.13	HELICAL INVOLUTE INTERPOLATION (G02.2,G03.3).....	98
4.14	SPLINE INTERPOLATION (G06.1).....	99
4.15	SPIRAL INTERPOLATION, CONICAL INTERPOLATION (G02,G03)	106
4.16	SMOOTH INTERPOLATION (G05.1)	115
4.17	NURBS INTERPOLATION (G06.2)	120
4.18	3-Dimensional Circular Interpolation (G02.4 and G03.4)	133
4.19	THREADING (G33)	137
4.20	INCH THREADING (G33)	140
4.21	CONTINUOUS THREADING (G33)	141
5	FEED FUNCTIONS	142
5.1	General	143
5.2	RAPID TRAVERSE.....	145
5.3	CUTTING FEED.....	146
5.4	OVERRIDE	152
5.4.1	Feedrate Override	152
5.4.2	Rapid Traverse Override	153
5.5	CUTTING FEEDRATE CONTROL.....	154
5.5.1	Exact Stop (G09, G61)Cutting Mode (G64)Tapping Mode (G63)	155
5.5.2	Automatic Corner Override	156
5.6	AUTOMATIC VELOCITY CONTROL	160
5.6.1	Automatic velocity control during involute interpolation	160
5.6.2	Automatic Velocity Contol During Polar Coordinate Interpolation	163
5.7	DWELL	165
6	REFERENCE POSITION	166
6.1	REFERENCE POSITION RETURN.....	167
6.2	FLOATING REFERENCE POSITION RETURN (G30.1)	171
7	COORDINATE SYSTEM	173
7.1	MACHINE COORDINATE SYSTEM	174
7.2	WORKPIECE COORDINATE SYSTEM	176
7.2.1	Setting a Workpiece Coordinate System (G92)	177
7.2.2	Setting Workpiece Coordinate System (G54 to G59)	178

7.2.3	Selecting Workpiece Coordinate System(G54 to G59)	180
7.2.4	Changing Workpiece Coordinate System	181
7.2.5	Adding Workpiece Coordinate Systems (G54.1)	184
7.2.6	Workpiece Coordinate System Preset (G92.1)	186
7.2.7	Automatically presetting the workpiece coordinate system	188
7.3	LOCAL COORDINATE SYSTEM	189
7.4	PLANE SELECTION	191
7.5	PLANE CONVERSION FUNCTION	192
8	COORDINATE VALUE AND DIMENSION	198
8.1	ABSOLUTE AND INCREMENTAL PROGRAMMING	199
8.2	POLAR COORDINATE COMMAND (G15,G16)	200
8.3	INCH/METRIC CONVERSION (G20,G21)	203
8.4	DECIMAL POINT INPUT/ POCKET CALCULATOR TYPE DECIMAL POINT INPUT	204
8.5	DIAMETER AND RADIUS PROGRAMMING	206
8.6	PROGRAMMABLE SWITCHING OF DIAMETER/RADIUS SPECIFICATION	207
9	SPINDLE SPEED FUNCTION (S FUNCTION)	210
9.1	SPECIFYING THE SPINDLE SPEED WITH A CODE	211
9.2	CONSTANT SURFACE SPEED CONTROL (G96, G97)	212
9.3	SPINDLE POSITIONING FUNCTION	217
9.3.1	Spindle Positioning	219
9.3.2	Orientation	220
9.3.3	Canceling the Spindle Positioning Mode	221
9.4	SPINDLE SPEED FLUCTUATION DETECTION (G26, G25)	223
10	TOOL FUNCTION (T FUNCTION)	229
10.1	TOOL SELECTION FUNCTION	230
10.2	TOOL LIFE MANAGEMENT FUNCTION	231
10.2.1	Tool Life Management Data	232
10.2.2	Register, Change and Delete of Tool Life Management Data	233
10.2.3	Tool Life Management Command in a Machining Program	236
10.2.4	Tool Service Life Count and Tool Selection	241
10.2.5	Tool Life Count Restart M Code	243
11	AUXILIARY FUNCTION	244
11.1	AUXILIARY FUNCTION (M FUNCTION)	245
11.2	MULTIPLE M COMMANDS IN A SINGLE BLOCK	247
11.3	THE AUXILIARY FUNCTIONS	248
12	PROGRAM CONFIGURATION	249
12.1	PROGRAM SECTION CONFIGURATION	251
12.2	SUBPROGRAM (M98, M99)	257
12.3	PROGRAM NUMBER	261

12.4 PROGRAM COMPONENTS OTHER THAN PROGRAM SECTIONS	262
12.5 EXTERNAL DEVICE SUBPROGRAM CALL (M198)	265
13 FUNCTIONS TO SIMPLIFY PROGRAMMING	267
13.1 CANNED CYCLE	268
13.1.1 High-speed Peck Drilling Cycle (G73)	274
13.1.2 Left-handed Tapping Cycle (G74)	276
13.1.3 Fine Boring Cycle (G76)	279
13.1.4 Drilling Cycle, Spot Drilling (G81)	282
13.1.5 Drilling Cycle Counter Boring Cycle (G82)	284
13.1.6 Peck Drilling Cycle (G83)	286
13.1.7 Tapping Cycle (G84)	288
13.1.8 Boring Cycle (G85)	291
13.1.9 Boring Cycle (G86)	293
13.1.10 Boring Cycle/Back Boring Cycle (G87)	295
13.1.11 Boring Cycle (G88)	300
13.1.12 Boring Cycle (G89)	302
13.1.13 Canned Cycle Cancel (G80)	304
13.1.14 Example of Canned Cycle	305
13.2 RIGID TAPPING	307
13.2.1 Rigid Tapping (G84.2)	308
13.2.2 Left-handed Rigid Tapping Cycle (G84.3)	311
13.2.3 Rigid tapping Orientation Function	314
13.2.4 Peck Rigid Tapping Cycle (G84 or G74)	316
13.2.5 Three-dimensional rigid tapping	318
13.3 EXTERNAL MOTION FUNCTION (G81)	320
13.4 OPTIONAL ANGLE CHAMFERING AND CORNER ROUNDING	321
13.5 PROGRAMMABLE MIRROR IMAGE (G50.1, G51.1)	325
13.6 INDEX TABLE INDEXING FUNCTION	327
13.7 FIGURE COPY (G72.1,G72.2)	330
13.8 NORMAL DIRECTION CONTROL (G40.1, G41.1, G42.1)	338
13.9 THREE-DIMENSIONAL COORDINATE CONVERSION (G68,G69)	342
14 COMPENSATION FUNCTION	352
14.1 TOOL LENGTH OFFSET (G43,G44,G49)	353
14.1.1 General	354
14.2 TOOL OFFSET(G45-G48)	357
14.3 OVERVIEW OF CUTTER COMPENSATION C (G40 - G42)	362
14.4 DETAILS OF CUTTER COMPENSATION C	369
14.4.1 General	370
14.4.2 Tool Movement in Start-up	374
14.4.3 Tool Movement in the Offset Mode	381
14.4.4 Tool Movement in Offset Mode Cancel	402
14.4.5 Overcutting by Cutter Compensation	410
14.4.6 Interference Check	414
14.4.7 Cutter Compensation by Input from MDI	429
14.4.8 Vector Holding (G38)	431
14.4.9 Corner Circular Interpolation (G39)	433
14.5 THREE-DIMENSIONAL TOOL COMPENSATION (G40, G41)	436
14.6 TOOL COMPENSATION VALUES	440

14.6.1	Tool Compensation Memory A	442
14.6.2	Tool Compensation Memory B	442
14.6.3	Tool Compensation Memory C	442
14.7	NUMBER OF TOOL COMPENSATION SETTINGS	443
14.8	CHANGING THE TOOL COMPENSATION AMOUNT.....	444
14.9	SCALING (G50,51).....	445
14.10	COORDINATE SYSTEM ROTATION (G68,G69)	451
14.11	TOOL OFFSETS BASED ON TOOL NUMBERS	457
14.11.1	Tool Data Registration, Modification, and Deletion	458
14.11.2	Tool Offset Based on Tool Numbers	460
14.11.3	Relationships with Other Functions	464
14.12	TOOL AXIS DIRECTION TOOL LENGTH COMPENSATION	466
14.13	ROTARY TABLE DYNAMIC FIXTURE OFFSET.....	473
14.14	THREE-DIMENSIONAL CUTTER COMPENSATION	480
14.14.1	Tool Side Compensation	481
14.14.2	Leading Edge Offset	496
14.15	DESIGNATION DIRECTION TOOL LENGTH COMPENSATION	504
15	PROGRAMMABLE PARAMETER INPUT (G10)	511
16	MEASUREMENT FUNCTION	513
16.1	SKIP FUNCTION (G31)	514
16.2	SKIPPING THE COMMANDS FOR SEVERAL AXES.....	517
16.3	HIGH SPEED SKIP SIGNAL (G31)	518
16.4	MULTISTAGE SKIP (G31.1 TO G31.4)	519
16.5	AUTOMATIC TOOL LENGTH MEASUREMENT (G37)	521
16.6	TORQUE LIMIT SKIP.....	525
17	CUSTOM MACRO	529
17.1	VARIABLES	530
17.2	SYSTEM VARIABLES	535
17.3	ARITHMETIC COMMANDS	564
17.4	MACRO STATEMENTS AND NC STATEMENTS	570
17.5	BRANCH AND REPETITION	571
17.5.1	Unconditional Branch (GOTO Statement)	571
17.5.2	Conditional Branch (IF Statement)	572
17.5.3	Repetition (While Statement)	574
17.6	MACRO CALL	577
17.6.1	Simple Call (G65)	578
17.6.2	Modal Call : Move Command Call (G66).....	584
17.6.3	Modal Call : Per-Block Call (G66.1)	587
17.6.4	Macro Call Using G Code	588
17.6.5	Macro Calls with G Codes (Specification of Multiple G Codes)	590
17.6.6	Macro Calls with G Codes with the Decimal Point (Specification of Multiple G Codes)	591
17.6.7	Macro Call Using an M Code	592
17.6.8	Macro Calls with M Codes with the Decimal Point (Specification of Multiple G Codes)	593

17.6.9 Subprogram Call Using an M Code	594
17.6.10 Subprogram Call Using an M Code (Specification of Multiple G Codes)	595
17.6.11 Subprogram Calls Using a T Code	596
17.6.12 Subprogram Calls Using a S Code	597
17.6.13 Subprogram Calls Using a 2nd Auxiliary Function Code	598
17.6.14 Sample Program	599
17.7 PROCESSING MACRO STATEMENTS	601
17.8 REGISTERING CUSTOM MACRO PROGRAMS	603
17.9 CODES AND RESERVED WORDS USED IN CUSTOM MACROS	604
17.10 WRITE-PROTECTING COMMON VARIABLES	606
17.11 DISPLAYING A MACRO ALARM AND MACRO MESSAGE IN JAPANESE	607
17.12 EXTERNAL OUTPUT COMMANDS	608
17.13 LIMITATIONS	613
17.14 INTERRUPTION TYPE CUSTOM MACRO	615
17.14.1 Specification Method	616
17.14.2 Details of Functions	618
18 HIGH-SPEED CUTTING FUNCTIONS	627
18.1 MULTIBUFFER (G05.1)	628
18.2 DECELERATION BASED ON ACCELERATION DURING CIRCULAR INTERPOLATION	632
18.3 ADVANCED PREVIEW CONTROL(G05.1)	635
18.4 LOOK-AHEAD ACCELERATION/DECELERATION BEFORE INTERPOLATION (G05.1)	636
18.5 FINE HPCC (G05.1)	638
18.6 MACHINING TYPE IN HPCC SCREEN PROGRAMMING (G05.1 OR G10)	641
19 AXIS CONTROL FUNCTIONS	643
19.1 AXIS INTERCHANGE	644
19.2 TWIN TABLE CONTROL	647
19.2.1 Tool Length Compensation in tool axis direction with Twin Table Control	651
19.3 SYNCHRONIZATION CONTROL	654
19.4 TANDEM CONTROL	655
19.5 CHOPPING FUNCTION (G80,G81.1)	656
19.6 PARALLEL AXIS CONTROL	663
19.7 ROTARY AXIS ROLL-OVER	668
19.8 MULTIPLE ROTARY CONTROL AXIS FUNCTION	670
19.9 ELECTRONIC GEAR BOX (G80, G81, G80.5, G81.5)	672
19.9.1 Command Specification (G80.5, G81.5)	673
19.9.2 Command Specification Compatible with Hobbing Machine (G80,G81)	675
19.9.3 Example of Controlled Axis Configuration	678
19.9.4 Sample Programs	679
19.9.5 Synchronization Ratio Specification Range	683
19.9.6 Retract Function	688
19.9.7 Electronic Gear Box Automatic Phase Synchronization	690

19.10 SKIP FUNCTION FOR EGB AXIS(G31.8)	694
19.11 TOOL WITHDRAWAL AND RETURN (G10.6)	696
APPENDIX	701
A TAPE CODE LIST	703
B LIST OF FUNCTION AND TAPE FORMAT	706
C RANGE OF COMMAND VALUE	711
D NOMOGRAPHS	715
D.1 INCORRECT THREADED LENGTH.....	716
D.2 SIMPLE CALCULATION OF INCORRECT THREAD LENGTH	718
D.3 TOOL PATH AT CORNER	720
D.4 RADIUS DIRECTION ERROR AT CIRCLE CUTTING.....	723
E TABLE OF KANJI AND HIRAGANA CODES	724
F ALARM LIST	732
F.1 PS ALARM (ALARMS RELATED TO PROGRAM)	733
F.2 BG ALARM (ALARMS RELATED TO BACKGROUND EDIT)	747
F.3 SR ALARM	749
F.4 SW ALARM (ALARMS RELATED TO PARAMETER WRITING)	751
F.5 SV ALARM (ALARMS RELATED TO SERVO)	752
F.6 OT ALARM	756
F.7 IO ALARM	758
F.8 PW ALARM (POWER MUST BE TURNED OFF THEN ON AGAIN)	758
F.9 SP ALARM (ALARMS RELATED TO SPINDLE)	759
F.10 OH ALARM (ALARMS RELATED TO OVERHEAT)	762