

NTC **V** series
**MACHINING CENTER
PROGRAMMING MANUAL**

FANUC - 11M Specification
FANUC - 0M

Table of Contents

Foreward.....	A - 1
---------------	-------

1. INTRODUCTION TO PROGRAMMING

1-1 Machining center processes.....	A - 2
1-2 Programming procedures.....	A - 3
1-3 Machine center preparation.....	A - 4

2. PREPARATIONS FOR NC PROGRAMMING

2-1 Preparing the work guide table.....	B - 1
2-2 Preparing the process sheet.....	B - 3
2-3 Programming precautions.....	B - 8
2-4 Determining the programming method	
2-5 Determining program zero.....	B - 17
2-6 The fundamentals of cutting movement.....	B - 18
2-7 Determining the cutting conditions.....	B - 20
2-8 Cutting work precautions.....	B - 21
2-9 Referring to calculations when programming.....	B - 23
2-10 Work guide table example.....	B - 25
2-11 Process sheet example.....	B - 26
2-12 Materials for the operator from the programmer.....	B - 27
2-13 Touring system.....	B - 29
2-14 Program check precautions.....	B - 30

3. PROGRAMMING

3-1 Coordinate and movement symbols.....	C-1
3-2 Basic addresses and programmable values.....	C-3
3-3 Basic information required for programming.....	C-4
3-3-1 Program tape.....	C-4
3-3-2 Tape code.....	C-4
3-3-3 Tape format.....	C-4
3-3-4 Elements of a block.....	C-4
3-3-5 Block construction example.....	C-5
3-3-6 Address.....	C-5
3-3-7 Word.....	C-5
3-3-8 Program number (0 ---).....	C-5
3-3-9 Sequence number (N ---).....	C-5
3-3-10 Preparatory Function (G Function).....	C-6
3-3-11 Dimension word.....	C-7
3-3-12 Feed Function (F Function).....	C-7
3-3-13 Spindle Function (S Function).....	C-10
3-3-14 Tool Function (T Function).....	C-11
3-3-15 Miscellaneous Function (M Function).....	C-13
3-3-16 Offset number.....	C-14
3-3-17 Dwell.....	C-15
3-3-18 Program number specification.....	C-15
3-3-19 Repetition.....	C-15
3-3-20 2nd Miscellaneous Function.....	C-15
3-3-21 Parameters.....	C-15
3-3-22 End of block.....	C-15
3-4 List of M, S and T codes.....	C-16
3-5 Basic programming parameters for TMC-V Series Machining Centers.....	C-20

3-6 Approach method for return to Z axis machine zero from Z axis part

zero and basic programming pattern for Z axis return to machine zero

.....C - 36

3-7 Preparatory Function (G Function).....C - 65

3-7-1 List of G codes.....C - 67

3-7-2 Concerning coordinate systems (G92, G54 ~ G59, G53, G52).....C - 69

3-7-3 Setting the coordinate system (G92).....C - 71

3-7-4 Absolute programming/incremental programming (G90, G91).....C - 73

3-7-5 Basic movements of the machine (G00, G01).....C - 75

3-7-6 Basic movements of the machine (G02, G03).....C - 79

3-7-7 Dwell (G04).....C - 84

3-7-8 Specifying the plane (G17, G18, G19).....C - 87

3-7-9 Concerning the reference point (G27, G28, G29, G30).....C - 88

3-7-10 Return to the reference point check (G27).....C - 89

3-7-11 Automatic return to the reference point (G28).....C - 91

3-7-12 Return from the reference point (G29).....C - 94

3-7-13 Automatic return to 2nd ~ 4th reference points (G30).....C - 96

3-7-14 Tool length compensation (G43, G44, G49).....C - 98

3-7-15 Tool position offset (G45, G46, G47, G48).....C - 106

3-7-16 Cutter compensation (G43, G44, G49).....C - 115

3-7-17 Workpiece coordinate system (G54 ~ G59).....C - 158

3-7-18 Programming the workpiece coordinate system offset distance

(G10).....C - 184

3-7-19 Local coordinate system (G52).....C - 187

3-7-20 Machine coordinate system (G53).....C - 193

3-7-21 Canned cycles (G73, G74, G80 ~ G89).....C - 199

3-7-22 Return point level (G98, G99).....C - 201

3-7-23 Exact stop check (G09).....C - 227

3-7-24 Exact stop check mode/exact stop check mode cancel (G61, G64).....C - 227

3-7-25 Helical cutting (G02, G03).....	C - 229
3-7-26 One-way position (G60).....	C - 230
3-7-27 Automatic corner override (G62).....	C - 232
3-7-28 Calling of custom macro (G65).....	C - 234
3-7-29 Programming of tool offset distance (H/D codes) (G10).....	C - 238

4. MAIN PROGRAMS AND SUBPROGRAMS

4-1 Main programs and subprograms.....	D - 1
4-2 Example of subprogramming.....	D - 3
4-3 Example of subprogramming L number of repetitions.....	D - 22

5. OTHER PROGRAMMING EXAMPLES

5-1 Concerning programming for true circle cutting.....	E - 1
5-2 Concerning programming using the 4th (A) axis.....	E - 4
5-3 Example programming using APC (Auto Pallet Changer).....	E - 14
5-4 Example programming using optional block skip (/).....	E - 15
5-5 Precautions for spindle start commands used during machining and example programming.....	E - 16
5-6 Programming using mirror image (M32, M33, M34, M35).....	E - 17
5-7 Programming of G92 command for different Z axis program zero.....	E - 22
5-8 Programming of back machining using circular interpolation.....	E - 23

6. APPENDIX

6-1 NC terminology and FANUC-11M reading abbreviations.....	F - 1
6-2 Basic programming examples for TMC-V Series Machining Centers	
6-2-1 Programming examples.....	F - 6
6-2-2 Drawing of boring and tap machining processes.....	F - 8
6-2-3 Programming solution examples 1 ~ 8.....	F - 9
6-2-4 Drawing of repetitive subprogram programming and circle pocket machining.....	F - 17
6-2-5 Programming solution examples 1 ~ 7.....	F - 18
6-3 Process sheet.....	F - 26
6-4 Tool compensation table.....	F - 27
6-5 Program zero settings table.....	F - 28