

**FT600 TURNING CENTERS
PROGRAMMING MANUAL
(FANUC SYSTEM 6T-B)**



CONTENTS

1. Functions	1
1-1 Preparatory Function (G Function)	1
1-2 Miscellaneous Function (M Function)	4
1-3 Spindle Speed Function (S Function)	12
1-4 Feed Function (F function)	14
1-5 Tool Function (T Function)	16
1-6 How to Use Functions	17
2. Fundamentals	21
2-1 Definition of Coordinate Axes	21
2-2 Increment System	21
2-3 Maximum Programmable Dimensions	22
2-4 Absolute and Incremental Programming	22
2-5 Decimal Point Programming	22
2-6 Program Number	23
2-7 Sequence Number	23
2-8 Optional Block Skip	23
2-9 Rapid Traverse	23
3. Program Format	24
3-1 Positioning (G00)	24
3-2 Linear Interpolation (G01)	24
3-3 Chamfering and Corner R	25
3-4 Circular Interpolation (G02, G03)	26
3-5 Thread Cutting (G32)	27
3-6 Dwell (G04)	28
3-7 Return to Reference Point (G27 ~ G30)	28
3-8 Programming of Absolute Zero Point (G50)	29
3-9 Inch/Metric Conversion (G20, G21)	29
3-10 Feed Rate Command Change Over (G68, G69)	29
3-11 Constant Surface Speed Control (G96, G97)	29
3-12 Stored Stroke Limit (G22, G23)	30
3-13 Tool Nose Radius Compensation (G40 ~ G42)	31
3-14 Offset Value Setting by Program (G10)	31
3-15 Canned Cycles A (G90, G92, G94)	32
3-16 Multiple Repetitive Cycles (G70 ~ G76)	34
3-17 Canned Cycles B (G80, G83 ~ G85, G87 ~ G89)	45

4.	Cutting Conditions	51
4-1	Turning	51
4-2	Milling	57
4-3	Drilling	62
4-4	Reaming and Tapping	63
4-5	Thrust	65
4-6	Surface Roughness	66
4-7	Cutting Speed	67
4-8	Relation between Workpiece Diameter and Feed Rate of Rotary Tool	68
4-9	Chucking Force	69