

TC-31A

TC-22A

TC-S2A

TC-32A

TC-R2A

TC-20A

TC-S2B

NC -PROGRAMMING MANUAL

Please read this manual carefully before starting operation.

brother

Chapter 1 Program Composition ----- 1-1

1.1	Types and composition of program -----	1-2
1.2	Composition of block -----	1-2
1.3	Composition of word -----	1-3
1.4	Numerical values -----	1-3
1.5	Sequence number -----	1-4
1.6	Optional block skip -----	1-4
1.7	Control out/in function -----	1-5

Chapter 2 Coordinate Command ----- 2-1

2.1	Coordinate system and coordinate value -----	2-2
2.2	Machine zero point and machine coordinate system -----	2-3
2.3	Working coordinate system -----	2-3

Chapter 3 Preparation Function ----- 3-1

3.1	Outline of G code -----	3-2
3.2	Positioning (G00) -----	3-7
3.3	Linear interpolation (G01) -----	3-7
3.3.1	Chamfering to desired angle and cornering C -----	3-9
3.4	Circular / helical interpolation (G02, G03) -----	3-12
3.4.1	Circular interpolation -----	3-12
3.4.1.1	XY Circular interpolation -----	3-12
3.4.1.2	XZ Circular interpolation -----	3-13
3.4.1.3	YZ Circular interpolation -----	3-14
3.4.2	Helical interpolation -----	3-18
3.5	Dwell(G04) -----	3-18
3.6	Exact stop check (G09, G61, G64) -----	3-19
3.7	Programmable data input (G10) -----	3-22
3.8	Soft limit -----	3-25
3.8.1	Stroke -----	3-25
3.8.2	Stroke limit -----	3-25
3.8.3	Programmable stroke limit (G22) -----	3-26
3.9	Return to the reference point (G28) -----	3-27
3.10	Return from the reference point (G29) -----	3-28

3.11	Return to the 2nd/3rd/4th reference point (G30)-----	3-28
3.12	Selection of machine coordinate system (G53)-----	3-29
3.13	Selection of working coordinate system (G54~G59) -----	3-29
3.14	Additional working coordinate system selection (G54.1)-----	3-29
3.15	Coordinate rotation function (G68,G69) -----	3-30
3.16	Coordinate rotation using measured results (G168)-----	3-31
3.17	Absolute command and incremental command (G90,G91) -----	3-31
3.18	Change of working coordinate system (G92) -----	3-33
3.19	Measurement feed (G131,G132) -----	3-35
3.20	Change of tap twisting direction (G133,G134)-----	3-36
3.21	Local coordinate system function(G52)-----	3-36
3.22	Single direction positioning function(G60)-----	3-37
3.23	G code priority -----	3-38

Chapter 4

Preparation Function (tool offset function)-- 4-1

4.1	Tool Dia Offset (G40,G41,G42) -----	4-2
4.1.1	Tool dia offset function -----	4-2
4.1.1.1	Tool dia fine compensation -----	4-2
4.1.2	Cancel Mode -----	4-3
4.1.3	Start-up -----	4-4
4.1.3.1	Inside cutting ($180^\circ \leq \theta$) -----	4-4
4.1.3.2	Outside cutting ($90^\circ \leq \theta < 180^\circ$) -----	4-5
4.1.3.3	Outside cutting ($\theta < 90^\circ$) -----	4-6
4.1.4	Offset Mode -----	4-7
4.1.4.1	Inside cutting ($180^\circ \leq \theta$) -----	4-7
4.1.4.2	Outside cutting ($90^\circ \leq \theta < 180^\circ$) -----	4-9
4.1.4.3	Outside cutting -----	4-10
4.1.4.4	Exceptional case -----	4-11
4.1.5	Offset Cancel -----	4-12
4.1.5.1	Inside cutting ($180^\circ \leq \theta$) -----	4-12
4.1.5.2	Outside cutting ($90^\circ \leq \theta < 180^\circ$) -----	4-13
4.1.5.3	Outside cutting ($\theta < 90^\circ$) -----	4-14
4.1.6	G40 single command -----	4-15
4.1.7	Change of offset direction in offset mode -----	4-16
4.1.8	Change of offset direction in offset mode -----	4-17

4.1.8.1	When there is a cross point -----	4-17
4.1.8.2	When there is no cross point -----	4-18
4.1.8.3	When offset path becomes more than a circle -----	4-19
4.1.9	G cord command for tool dia offset in offset mode -----	4-20
4.1.10	Notes on tool dia offset -----	4-21
4.1.11	Override function related to tool dia offset -----	4-27
4.1.11.1	Automatic corner override -----	4-27
4.1.11.2	Override of the inside circular cutting -----	4-28
4.2	Tool length offset (G43,G44,G49) -----	4-29
4.2.1	Tool length fine offset -----	4-30

Chapter 5

Preparation Function (canned cycle) ----- 5-1

5.1	List of canned cycle function -----	5-2
5.2	Basic motions in canned cycle -----	5-4
5.3	General description of canned cycle -----	5-5
5.3.1	Command related to canned cycle motions -----	5-5
5.3.2	Setting of data in absolute/incremental command -----	5-5
5.3.3	Types of return point (G98,G99) -----	5-6
5.3.4	Canned cycle motion conditions -----	5-6
5.3.5	Machining data of canned cycle -----	5-7
5.3.6	Repeat number of canned cycle -----	5-8
5.4	Details of Canned Cycle -----	5-9
5.4.1	High-speed peck drilling cycle (G73) -----	5-9
5.4.2	Reverse tapping cycle (G74) -----	5-10
5.4.3	Fine boring cycle (G76) -----	5-11
5.4.4	Tapping cycle (G77) -----	5-12
5.4.5	Reverse tapping cycle (synchro mode) (G78) -----	5-13
5.4.6	Drilling cycle (G81,G82) -----	5-14
5.4.7	Peck drilling cycle (G83) -----	5-16
5.4.8	Tapping cycle (G84) -----	5-17
5.4.9	Boring cycle (G85,G89) -----	5-18
5.4.10	Boring cycle (G86) -----	5-19
5.4.11	Back boring cycle (G87) -----	5-21
5.4.12	High speed peck drilling cycle (G173) -----	5-22
5.4.13	End mill tap cycle (G177) -----	5-23
5.4.14	End mill tap cycle (G178) -----	5-24

5.4.15	Double drilling cycle (G181,G182) -----	5-25
5.4.16	Peck drilling cycle (G183) -----	5-26
5.4.17	Double boring cycle (G185,G189) -----	5-27
5.4.18	Double boring cycle (G186) -----	5-28
5.4.19	Canned cycle of reducing step -----	5-29
5.4.20	Canned cycle cancel (G80) -----	5-35
5.4.21	Notes on canned cycle -----	5-36
5.5	Canned cycle for tool change (non-stop ATC)(G100) -----	5-37

Chapter 6

Preparation Function (coordinate calculation) 6-1

6.1	List of coordinate calculation function -----	6-2
6.2	Coordinate calculation parameter -----	6-2
6.3	Details of coordinate calculation function -----	6-3
6.3.1	Bolt hole circle -----	6-3
6.3.2	Linear (angle) -----	6-4
6.3.3	Linear (X,Y) -----	6-4
6.3.4	Grid -----	6-5
6.4	Usage of coordinate calculation function -----	6-6

Chapter 7 Macro ----- 7-1

7.1	What is a Macro? -----	7-2
7.2	Variable Function -----	7-3
7.2.1	Outline of variable function -----	7-3
7.2.2	Expression of variable -----	7-3
7.2.3	Undefined variable -----	7-4
7.2.4	Types of variables -----	7-5
7.2.5	Variable display and setting -----	7-6
7.2.6	Macro (system variable) -----	7-6
7.3	Calculation Function -----	7-11
7.3.1	Calculation type -----	7-11
7.3.2	Calculation order -----	7-12
7.3.3	Precautions for calculation -----	7-12
7.4	Control Function -----	7-13

7.4.1	GOTO statement (unconditional branch) -----	7-13
7.4.2	IF statement (conditional branch)-----	7-14
7.4.3	WHILE statement (repetition) -----	7-15
7.4.4	Precautions for control function -----	7-16
7.5	Call Function -----	7-18
7.5.1	Simple call function -----	7-19
7.5.2	Modal call function -----	7-20
7.5.3	Argument for macro calls -----	7-21
7.5.4	Difference between G65 and M98 -----	7-23
7.5.5	Multiple nesting call -----	7-24

Chapter 8 Z axis measurement system ----- 8-1

8.1	Cautions -----	8-2
8.2	Setting -----	8-3
8.3	Z-axis thermal displacement offset -----	8-6
8.3.1	Offset capacity -----	8-6
8.3.2	For correct thermal displacement offset -----	8-7
8.3.3	Command -----	8-8
8.3.3.1	Determination of thermal measurement -----	8-8
8.3.3.2	Thermal measurement operation -----	8-9
8.3.4	Program examples -----	8-10
8.3.5	Thermal measurement data screen -----	8-11
8.3.6	Outputting thermal displacement measurement data to the computer -----	8-12
8.3.7	Example parameter setting and alarm -----	8-13
8.3.8	Example offset -----	8-14
8.4	Tool breakage detection -----	8-20
8.4.1	Determination of tool breakage detection -----	8-20
8.4.2	Program examples -----	8-21
8.5	Automatic tool length measurement -----	8-23
8.5.1	Command -----	8-23
8.5.2	Operation -----	8-25

Chapter 9 Origin measurement ----- 9-1

9.1	Before origin measurement -----	9-4
9.2	Setting of data on origin measurement -----	9-4

9.3	Operation of origin measurement -----	9-8
9.3.1	Corner -----	9-8
9.3.2	Parallel -----	9-12
9.3.3	Circle -----	9-15
9.3.4	Z level -----	9-19
9.3.5	Positioning to the measurement position -----	9-19
9.4	Display of the measured results -----	9-20
9.5	Lock key operations -----	9-22
Chapter 10	Subprogram function -----	10-1
10.1	Making subprogram -----	10-2
10.2	Simple call -----	10-3
10.3	Return No. designation from sub program -----	10-4
10.3.1	Command by sub program -----	10-4
10.3.2	Command by main program -----	10-4
Chapter 11	Feed function -----	11-1
11	Feed function -----	11-2
Chapter 12	S,T,M function -----	12-1
12.1	S function -----	12-2
12.2	T function -----	12-2
12.2.1	Commanded by tool No. -----	12-2
12.2.2	Commanding by pot No. (magazine No.) -----	12-2
12.2.3	Commanded by group No. -----	12-2
12.3	M function -----	12-4
12.3.1	Program stop (M00) -----	12-9
12.3.2	Optional stop (M01) -----	12-9
12.3.3	End of program (M02, M30) -----	12-9
12.3.4	Commands on the spindle (M03, M04, M05, M19, M111) -----	12-10
12.3.4.1	Spindle orientation to desired angle (M19) -----	12-10
12.3.5	M signal level output (M400~M409) -----	12-10
12.3.6	Tool change (M06) -----	12-10
12.3.7	Workpiece counter specification (M211~M214) -----	12-23
12.3.8	Workpiece counter cancel (M221~M224) -----	12-23

12.3.8.1	Tool life counter -----	12-24
12.3.9	Automatic corner deceleration (M232, M233) -----	12-24
12.3.10	M120 function -----	12-24
12.3.11	Tool breakage detection (M200 and M201) -----	12-24
12.3.12	Tap time constant selection (M241 to 250) -----	12-24
12.3.13	Pallet related M codes (M410, M411, M430, and M431) -----	12-26
12.3.14	Unclamping and clamping C axis (M440 and M441) -----	12-26
12.3.15	Unclamping and clamping B axis (M440 and M441) -----	12-26
12.3.16	Unclamping and clamping Aaxis (M442 and M443)-----	12-26
12.3.17	One-shot output (M450, M451, M455, and M456) -----	12-27
12.3.18	Waiting until response is given (M460 to M469) -----	12-27

Chapter 13 Option ----- 13-1

13.1	Memory capacity expansion -----	13-2
13.2	Programming precautions when using rotation axis (index table) ---	13-2