

# **Cincom A20**

(2F7PL)

## **Programmer's Manual**

# Programmer's Manual

## 0. Table of Contents

This contents does not go into details.

Please refer to the table of contents for detailed headline listed on the top of each chapter.

0. Table of Contents.....	0-1
1. Outline.....	1-1
1.1 Outline of the Manual.....	1-3
1.2 Audience.....	1-4
2. Safety Precautions .....	2-1
2.1 Safety Signs .....	2-3
2.2 Emergency Stop Button.....	2-7
2.3 Safety Devices.....	2-8
2.4 Specifications for Safe Operation .....	2-13
2.5 General Precautions During Operation .....	2-14
2.6 Safety During Installing the Machine.....	2-15
2.7 Before Starting the Machine .....	2-16
2.8 Safety During Setup.....	2-17
2.9 Safety During Automatic Operation.....	2-18
2.10 Safety During Maintenance.....	2-19
2.11 Safety After Everyday Operation.....	2-19
2.12 Fire Prevention Practices.....	2-20
2.13 Handling Emergency Situations .....	2-22
2.13.1 Machine operation at occurrence of an accident.....	2-22
2.13.2 Emergency situations requiring evacuation.....	2-22
2.13.3 Power failure.....	2-22
2.13.4 Resuming work.....	2-22
2.14 Notes for prevention of machine damage .....	2-23

Code No.	C-A220PL VII 2E1-0003	MFG No.	A220/0007 ~	Issue Date	2008.7
----------	--------------------------	---------	-------------	------------	--------

3. Fundamentals of Programming .....	3-1
3.1 What is NC Programming? .....	3-3
3.1.1 Difference between cam control and NC .....	3-4
3.2 Process of Program Creation.....	3-5
3.3 NC Program Structure .....	3-7
3.3.1 Machining data (screen display).....	3-10
3.4 Drive Axis .....	3-12
3.4.1 Drive axis .....	3-12
3.4.2 Stroke Diagram.....	3-13
3.5 Coordinate System .....	3-16
3.5.1 Coordinate axes.....	3-16
3.5.2 Diameter designation .....	3-16
3.5.3 Signs and coordinate values .....	3-17
3.5.4 Coordinate system setup .....	3-17
3.5.5 Absolute and incremental commands .....	3-18
3.6 Machining Patterns .....	3-19
3.6.1 Machining Pattern List .....	3-20
3.6.2 Machining pattern flow .....	3-21
3.6.3 Free pattern (machining pattern cancel) G600.....	3-22
3.6.4 Front/back parallel machining G630.....	3-24
3.6.5 Pick-off, center support G650 .....	3-26
3.6.6 Front/back simultaneous machining G660 .....	3-29
3.6.7 Superimpose control .....	3-31
4. Command Codes .....	4-1
4.1 Addresses Used in NC ProgramsNC .....	4-3
4.2 T Functions (Tool Selection) .....	4-4
4.3 T Code Arguments .....	4-7
4.3.1 T code arguments of gang tool post and rotary tools (T0100 to T0900).....	4-7
4.3.2 T Code arguments of front drills on gang tool post (T1100 to T1400) .....	4-17
4.3.3 Back drills on gang tool post (T5100 to T5400).....	4-21
4.3.4 Back spindle (T3000) .....	4-27
4.3.5 Back drills on back tool post (T3100 to T3400) .....	4-28
4.3.6 T codes and arguments effective in machining patterns .....	4-33
4.4 G Functions (G codes).....	4-35
4.4.1 List of G codes .....	4-35
4.4.2 Positioning through rapid feed (G00) .....	4-37
4.4.3 Linear interpolation (G01) .....	4-39
4.4.4 Circular interpolation (G02, G03) .....	4-41
4.4.5 Dwell (G04).....	4-43
4.5 F Functions (F Code) (Cutting Feedrate) .....	4-44
4.6 M Functions (M codes) .....	4-46
4.6.1 List of M codes.....	4-46
4.6.2 Spindle rotation (M3, M4, M5) .....	4-49
4.6.3 Back spindle rotation (M23, M24, M25).....	4-50

4.6.4	Tool spindle rotation (M80, M81, M82).....	4-51
4.6.5	Spindle chuck open/close (M6, M7, M700).....	4-52
4.6.6	Back spindle chuck open/close (M15, M16) .....	4-53
4.6.7	Move basket forward (M320, M780).....	4-54
4.6.8	Workpiece conveyor start (M31) .....	4-58
4.6.9	Knock-out advance/retract (M10, M11).....	4-58
4.6.10	Gang tool post retract (M151) .....	4-60
4.6.11	Thread Cutting and Chamfering ON/OFF (M204, M205) .....	4-61
4.6.12	Interference Check Function (M88, M89).....	4-63
4.7	S Functions (S codes) .....	4-64
4.8	! (Exclamation) Function (Queuing Code).....	4-65
5.	Sample Programming [Basic].....	5-1
5.1	Program Structure .....	5-3
5.1.1	Program structure chart .....	5-4
5.1.2	Guidelines for Designing a Machining Program .....	5-5
5.2	Preparation Processes .....	5-6
5.3	Machining Process .....	5-8
5.3.1	Front turning process .....	5-9
5.3.2	Rear turning process.....	5-10
5.3.3	Grooving process.....	5-11
5.3.4	Drilling process .....	5-12
5.4	Center Hole Machining .....	5-13
5.4.1	Spindle speed .....	5-13
5.4.2	Programmed point and cutting point (positioning) .....	5-13
5.4.3	Center machining.....	5-14
5.5	Deep Hole Drilling .....	5-15
5.6	Machining with Tap and Die.....	5-17
5.7	Tapping .....	5-18
5.8	Threading with Tool .....	5-19
5.8.1	Programmed point and cutting point .....	5-19
5.8.2	G92 threading canned cycle .....	5-20
5.8.3	Threading with G92.....	5-21
5.8.4	G92 taper threading canned cycle .....	5-23
5.9	Boring (Coordinate System Shift) .....	5-24
5.9.1	Boring Tool Signs .....	5-25
5.9.2	Diameter direction coordinate system shift and shift cancel.....	5-26
5.9.3	Longitudinal coordinate system shift and shift cancel.....	5-27
5.9.4	Diameter/longitudinal direction coordinate system shift and shift cancel .....	5-28
6.	Sample Programming [Advanced].....	6-1
6.1	Tool Position Offset .....	6-3
6.1.1	Examples of offset specification.....	6-4
6.2	Automatic Bar Feeder.....	6-7
6.2.1	M108 (Material change using an argument).....	6-7
6.2.2	General-purpose Material Change Program (by user program) .....	6-14

6.3	Subprograms .....	6-16
6.4	Spindle Speed Fluctuation Detection Function (G25, G26, M94, M95, M96, M97) .....	6-19
6.5	Constant Surface Speed Control (Main and Back Spindle) .....	6-21
6.6	Tool Nose R Compensation .....	6-23
6.7	Cut-off Tool Breakage Detection (M51) .....	6-27
6.8	Front/Back End-face Rigid Tapping (G84, G80) .....	6-31
6.9	Corner Chamfering/Rounding .....	6-34
6.10	Canned Drilling Cycle .....	6-36
6.11	Multiple Repetitive Cycle .....	6-39
6.12	Direct Input of On-drawing Dimensions .....	6-43
6.13	Continuous Threading .....	6-46
6.14	Threading (G32) .....	6-48
6.15	Longitudinal Cut-Off Cycle (G75) — Option .....	6-50
6.16	Link-Thread Machining .....	6-52
6.17	Multi-thread Cutting .....	6-55
6.18	Custom Macro .....	6-58
6.19	Medium Pressure Coolant Discharge Device .....	6-63
6.20	Setting the Free Tool Layout Pattern .....	6-64
6.20.1	Angle Setting .....	6-64
6.20.2	Tool setting .....	6-67
6.20.3	Standard tool pattern data .....	6-67
6.21	Machine Coordinate Value Specification (G53) .....	6-70
6.22	Y-axis Holder .....	6-71
6.23	Program Pre-analysis Function .....	6-73
6.23.1	Restrictions .....	6-73
6.23.2	Starting program pre-analysis function .....	6-75
6.23.3	Enabling or disabling program pre-analysis function .....	6-76
7	Sample Programming [Secondary Machining] .....	7-1
7.1	Secondary Machining .....	7-3
7.1.1	Difference between Turning and Secondary Machining .....	7-3
7.1.2	Machining Process Order .....	7-3
7.1.3	Coordinates for Secondary Machining .....	7-4
7.2	Main Spindle Indexing .....	7-5
7.2.1	Main Spindle Indexing (M28, M20) .....	7-5
7.2.2	Back Spindle Indexing (M78, M79) .....	7-7
7.2.3	Main Spindle C Axis (M18, M20) .....	7-8
7.2.4	Back Spindle C Axis (M48, M79) .....	7-10
7.2.5	Tool spindle rotation .....	7-11
7.3	Calculating the Command Speed .....	7-12
7.3.1	Calculating the Spindle Command Speed .....	7-12
7.3.2	Calculating the Tool Feedrate .....	7-13
7.4	Partial Program for 2-side Width Machining (Including Indexing) .....	7-14
7.5	Through-hole Machining .....	7-16
7.6	Circular Interpolation .....	7-17

7.6.1	Selecting the Plane .....	7-17
7.6.2	Offsetting the Tool Diameter .....	7-18
7.7	Cross Rigid Tapping (G88, G80) .....	7-21
7.7.1	Rigid tapping for outer circumference with a rotary tool (G88, G80).....	7-21
7.8	End Face Drilling Spindle (BSE107) .....	7-24
7.9	Milling Interpolation .....	7-26
7.9.1	Milling coordinates .....	7-26
7.9.2	Milling plane .....	7-27
7.9.3	Programming of a milling process .....	7-28
7.9.4	Calculation of milling coordinates .....	7-30
7.9.5	Program format .....	7-35
7.9.6	Milling compensation .....	7-36
7.9.7	List of NC unit errors during milling interpolation .....	7-37
7.9.8	Example of using the milling interpolation function 1 (D cut) .....	7-38
7.9.9	Example of using the milling interpolation function 2 .....	7-40
7.9.10	Example of using the milling interpolation function 3 .....	7-42
7.9.11	Example of using the milling interpolation function 4 .....	7-45
7.9.12	Example of using the milling interpolation function 5 .....	7-47
7.9.13	Example of using the milling interpolation function 6 .....	7-49
8 .	Sample Programming [Back Machining] .....	8-1
8.1	Back Machining Program .....	8-3
8.2	Back Machining Coordinates .....	8-4
8.3	Back Machining Process .....	8-5
8.4	Back Spindle Air Blow (M72, M73) .....	8-7
8.5	Move Back Spindle to Waiting Point (M241) .....	8-7
8.6	Knock-out Advance/Retract (M10, M11) .....	8-8
8.7	Product Collection (M33, M34) .....	8-9
8.7.1	Pick-off collection mode .....	8-9
8.7.2	Cut-off and product collection .....	8-13
8.7.3	Basket-collection mode .....	8-15
8.8	Spindle Synchronization Control (G814, G813, G114.1, G113, M77) .....	8-18
8.9	Z1-Z2 Axis Sumperimposition (M130, M131) .....	8-20
8.10	C1-C2 Axis Sumperimposition (M190, M191) .....	8-22
8.11	Pick-off and Ending Process .....	8-24
8.11.1	Pick-off process .....	8-26
8.11.2	Re-chucking of a Long Workpiece .....	8-28
8.12	Executing the Last Program (G999) .....	8-33
8.13	Back Pick-Off Failure Detect (M50) .....	8-35
8.14	Phase Adjustment of Non-conform Material .....	8-36
8.14.1	Non-conformed material phase adjustment on Preparation screen .....	8-36
8.14.2	Non-conformed material phase adjustment in MDI mode .....	8-38
8.15	Machining with Tool or Rotary Tool in Back Machining .....	8-39
8.16	Back Spindle Feed Per Rotation ON/OFF (G44/G43) .....	8-42
8.17	Stroke and Notes on Machining .....	8-43

8.18	Use of Machining Data "Back Long-neck Chuck Protrusion Length" .....	8-45
8.19	Setting Acceleration/Deceleration Time Constant in Rapid Feed ON/OFF (M360/M361) .....	8-48
9	Programming Practices .....	9-1
9.1	Programming Practice 1 (Front Machining).....	9-3
9.2	Programming Practice 2 .....	9-6
9.3	Programming Practice 3 (Back Machining).....	9-10
10	Cutting Conditions .....	10-1
10.1	Cutting Speeds and Feedrates .....	10-3
10.2	Thread Cutting Count with Tool .....	10-7
10.3	Cutting Condition Tables for the Secondary Machining Process.....	10-8
10.4	Quick Reference Table for Cutting Speed .....	10-9
11	Tooling .....	11-1