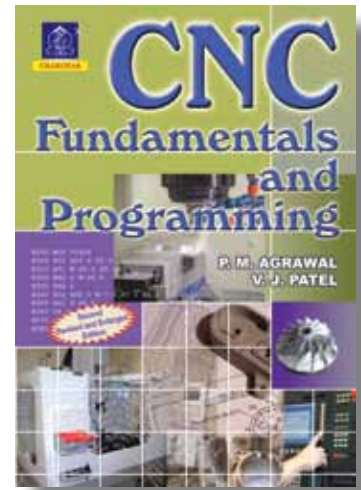


CNC FUNDAMENTALS AND PROGRAMMING



By
P. M. Agrawal, V. J. Patel

Edition : 2nd Edition : 2014
ISBN : 978-93-80358-87-1
Size : 170 mm × 240 mm
Binding : Paperback with 4 Color Jacket Cover
Pages : 292 + 16



₹ 200.00

ABOUT THE BOOK

This text-book explains the fundamentals of NC/CNC machine tools, operations and part programming which form essential portion of course on Computer Aided Manufacturing (CAM). This book also covers advanced topics such as Macro programming, DNC and Computer Aided Part Programming (CAPP) in detail.

In this second revised and enlarged edition, all the chapters are reviewed and relevant topics, examples, part programs, sketches, review questions and exercises have been added to enhance the utility of the book.

This book is divided into 6 major areas.

Chapter 1 to 4 cover the history, fundamentals and structure of NC/CNC machine tools.

Chapter 5, 6 and 7 cover turning center programming in detail. Various turning canned cycles are discussed in depth with the help of illustrative examples.

Chapter 8 and 9 are devoted to machining center programming. Drilling canned cycles are discussed in detail with relevant examples.

Chapter 10 and 11 cover advanced topics of subprogramming and macro programming.

Chapter 12 deals with the communications and networking of NC/CNC machine tools.

Chapter 13 and 14 explain the advanced programming using APT and CAD/CAM based programming respectively.

The book contains:

- * 258 Self explanatory and neatly drawn drawings
- * 62 Solved part programming examples
- * 37 Part programming exercises
- * 173 Review questions at the end of all the chapters
- * 101 Multiple choice questions.

It is the fervent hope of the authors that book will satisfy the needs of the Mechanical, Production, Mechatronics and Automobile Engineering students preparing for the B.Tech/B.E. examinations of all the Indian Universities, Diploma examinations conducted by various Boards of Technical Education, Certificate course as well as for the A.M.I.E., U.P.S.C., G.A.T.E., I.E.S. and other similar competitive and professional examinations. It should also be of an immense help to the practising Mechanical Engineers.

CONTENT

- 1 : INTRODUCTION TO NC/CNC MACHINE TOOLS
- 2 : BASICS OF NC/CNC MACHINE TOOLS
- 3 : CNC MACHINE TOOLS – STRUCTURE AND ELEMENTS
- 4 : BASICS OF CNC PROGRAMMING
- 5 : BASICS OF TURNING CENTER PROGRAMMING
- 6 : SINGLE PASS CANNED CYCLE FOR TURNING CENTERS
- 7 : MULTIPASS CANNED CYCLES IN TURNING
- 8 : BASICS OF MACHINING CENTER PROGRAMMING
- 9 : DRILLING CANNED CYCLES IN MACHINING CENTER
- 10: SUBPROGRAMMING
- 11: MACRO PROGRAMMING
- 12: DNC AND DATA COMMUNICATION
- 13: APT PROGRAMMING
- 14: CAD/CAM INTEGRATION
- APPENDIX I : MULTIPLE CHOICE QUESTIONS
- APPENDIX II : NC/CNC MANUAL PART PROGRAMMING CODES

INDEX

Checklist



Charotar Publishing House Pvt. Ltd. Opposite Amul Dairy, Civil Court Road, Post Box No. 65, ANAND 388 001 India
Telephone: (02692) 256237, Fax: (02692) 240089, e-mail: charotar@cphbooks.com, Website: www.cphbooks.in

CNC FUNDAMENTALS AND PROGRAMMING
DETAILED CONTENTS

Chapter 1 INTRODUCTION TO NC/CNC MACHINE TOOLS

- 1-1. History of NC machines
- 1-2. Introduction to numerical control (NC) machine tools
Definition
- 1-3. CNC (Computer Numerical Control)
Machine control unit
- 1-4. Differences between NC and conventional machine tools
 - (1) Spindle drives
 - (2) Guideways
 - (3) Motion transmission
 - (4) Feed drives
 - (5) Machine tool structure
 - (6) Control unit
 - (7) Feedback unit
 - (8) Automatic tool changer (ATC)
 - (9) Automatic pallet changer (APC)
- 1-5. Advantages of CNC over NC
 - (1) Controller
 - (2) Memory
 - (3) Part program editing
 - (4) Cutter compensation
 - (5) Multipass canned cycles
 - (6) Advance part programming
 - (7) Tool path simulation
 - (8) Conversational programming
 - (9) Diagnostic
- 1-6. Advantages of NC and CNC over Conventional machine tools
 - (1) Increased flexibility
 - (2) More complex geometry
 - (3) Higher production rate
 - (i) Reduced set-ups
 - (ii) Reduced lead time
 - (iii) Reduced non-machining time
 - (4) Higher accuracy and repeatability
 - (5) Reduced inspection
 - (6) Reduced operator skill
 - (7) Reduced scrap
 - (8) Reduced work-in-process inventory
 - (9) Elimination of profile tools
 - (10) Simpler fixtures
 - (11) Optimum cutting conditions
- 1-7. Limitations of NC/CNC machine tools
 - (1) Cost
 - (2) High maintenance costs
 - (3) Not cost effective for low production levels
 - (4) Programming skill
- 1-8. Specifications of a typical turning center
 - (1) Axes system
 - (2) Main spindle
 - (3) Tool turret
 - (4) Tail stock (with hydraulic quill)
 - (5) Other Features
- 1-9. Specifications of a typical vertical machining center (VMC)
 - (1) Axes system
 - (2) Table
 - (3) Headstock
 - (4) ATC system
 - (5) Special features
- 1-10. Role of NC/CNC technology in modern manufacturing
 - (1) Machining
 - (2) Unconventional machining
 - (3) Fabrication and welding
 - (4) Press work
 - (5) Material handling and assembly
 - (6) Inspection and measurement
 - (7) Wood working
- 1-11. Multi-tasking machine tools
Review questions 1

Chapter 2 BASICS OF NC/CNC MACHINE TOOLS

- 2-1. Axes designation
- 2-1-1. Coordinate system
- 2-1-2. Designating the axes
- 2-2. Axes system of different machine tools

- 2-2-1. Turning center
- 2-2-2. Vertical machining center (VMC)
- 2-2-3. Horizontal machining center
- 2-2-4. Turn mill center
- 2-2-5. Three axes planer
- 2-2-6. Four axes machining center
- 2-2-7. Five axes horizontal boring machine
- 2-2-8. Five axes vertical machining center
- 2-3. Reference points
 - 2-3-1. The grid system
 - 2-3-2. Machine origin or machine zero
 - 2-3-3. Part origin or part zero
 - 2-3-4. Tool reference point
- 2-4. CNC control systems
 - 2-4-1. Point-to-Point control (P-type)
 - (1) Axial path
 - (2) 45° path
 - (3) Linear Path
 - 2-4-2. Line control/straight cut control (L-type)
 - 2-4-3. Continuous path control (C-type)
 - (1) 2-D contouring control
 - (2) 2½-D contouring control
 - (3) 3-D contouring control
 - (4) 4-D contouring control
 - 2-4-4. Example of P, L and C type controls
- 2-5. Automatic tool changer (ATC)
- 2-6. Automatic pallet changer
Review questions 2

Chapter 3 CNC MACHINE TOOLS – STRUCTURE AND ELEMENTS

- 3-1. Machine tool structure
- 3-2. Machine tool spindle
 - (1) Belt driven spindle
 - (2) Integral motor-spindle
- 3-3. Spindle bearings
- 3-4. Guideways
- 3-5. Transmission system
 - 3-5-1. Recirculating ballscrews
 - (1) Double nut preloading
 - (2) Single nut preloading
 - 3-5-2. Roller screw
- 3-6. Drives
 - 3-6-1. Stepper motor
 - 3-6-2. AC/DC rotary servo motor
 - 3-6-3. Linear motor
- 3-7. Open loop and closed loop control systems
 - 3-7-1. Open loop control system
 - 3-7-2. Closed loop control system
- 3-8. Feedback devices
 - 3-8-1. Incremental optical encoders
 - 3-8-2. Absolute optical encoders
 - (1) Pure binary code
 - (2) Gray code
 - (3) Gray excess code
- 3-9. CNC Tooling
 - 3-9-1. Qualified tool
 - 3-9-2. Preset tools/semi-qualified tools
 - 3-9-3. Tool presetters
 - 3-9-4. Tool holders
Review questions 3
Exercise 3

Chapter 4 BASICS OF CNC PROGRAMMING

- 4-1. Coding systems
 - (1) Fixed sequential format
 - (2) Tab sequential format
 - (3) Word address format
 - (i) Alphabet
 - (ii) Digits
 - (iii) Operators
 - (iv) Special characters
- 4-2. Punched paper tapes
 - (1) ISO code
 - (2) EIA code
- 4-3. Punched tape readers



CNC FUNDAMENTALS AND PROGRAMMING
DETAILED CONTENTS

- (1) Electromechanical tape reader
- (2) Photoelectric tape reader
- (3) Pneumatic tape reader
- (4) Magnetic tapes
- 4-4. Types of codes
 - (1) Sequence or block number
 - (2) Preparatory code
 - (3) Miscellaneous code
 - (4) Axis words
 - (5) Feed word
 - (6) Speed word
 - (7) Tool number
- 4-4-1. Preparatory codes
- 4-4-2. Miscellaneous codes
- 4-4-3. Modal and Non-modal codes
- 4-5. Decimal point programming
- 4-6. Absolute and incremental positioning
- 4-6-1. Absolute programming or positioning
- 4-6-2. Incremental/relative programming or positioning
- 4-7. Diameter and radius programming
- 4-8. Programming functions
- 4-8-1. Data input units
- 4-8-2. Spindle speed control
 - (1) Direction
 - (2) Spindle speed
- 4-8-3. Feedrate control
- 4-8-4. Coordinate system preset
- 4-8-5. Tool and tool offset
- 4-8-6. Program control
 - (1) M00: Program stop
 - (2) M01: Optional stop
 - (3) M02: End of program
 - (4) M30: Program stop and rewind
- Review questions 4

Chapter 5 BASICS OF TURNING CENTER PROGRAMMING

- 5-1. Rapid traverse (G00)
- 5-2. Linear interpolation (G01)
- 5-3. Circular interpolation (G02/G03)
 - (1) Cutting direction (G02 or G03)
 - (2) Arc end point coordinates
 - (3) Radius of arc or center of arc
- 5-4. Tool compensations
 - (1) Geometry offsets
 - (2) Wear offsets
 - (3) Tool nose radius compensation
- 5-4-1. Geometry offsets
- 5-4-2. Wear offset
- 5-4-3. Tool nose radius compensation
- 5-5. Automatic reference point return (G28)
- 5-6. Dwell
- Review questions 5
- Exercise 5

Chapter 6 SINGLE PASS CANNED CYCLE FOR TURNING CENTERS

- 6-1. Rectangular turning cycle (G90)
- 6-2. Thread cutting cycle (G92)
- 6-3. Face turning cycle (G94)
- Review questions 6
- Exercise 6

Chapter 7 MULTIPASS CANNED CYCLES IN TURNING

- 7-1. Rough turning cycle (G71)
- 7-2. Face roughing cycle (G72)
- 7-3. Pattern repeating cycle (G73)
- 7-4. End face peck drilling cycle (G74)
- 7-5. Outer diameter grooving cycle (G75)
- 7-6. Multi-pass thread cutting cycle (G76)
- Review questions 7
- Exercise 7

Chapter 8 BASICS OF MACHINING CENTER PROGRAMMING

- 8-1. Rapid traverse (G00)
- 8-2. Linear interpolation (G01)
- 8-3. Circular interpolation (G02/G03)
- 8-3-1. Circular interpolation for arcs greater than 180°
- 8-3-2. Programming of full circle in one block
 - (1) Radius of arc programming
 - (2) Center of arc programming
- 8-4. Profile milling
- 8-5. Cutter compensations
- 8-5-1. Cutter radius compensation
 - (1) Offset plane selection and offset vector
 - (2) Cutter radius compensation to the left (G41)
 - (3) Cutter radius compensation to the right (G42)
 - (4) Ramp on move
 - (5) Ramp off move
 - (6) Tool movements in offset mode
 - (7) Change of the cutter compensation value
- 8-5-2. Tool length compensation
- 8-5-3. Tool wear compensation
- 8-6. Automatic reference point return (G28)
- Review questions 8
- Exercise 8

Chapter 9 DRILLING CANNED CYCLES IN MACHINING CENTER

- 9-1. Syntax of drilling canned cycles
 - (1) Return to the initial level (G98)
 - (2) Return to R level (G99)
- 9-2. Cancel the drilling canned cycle (G80)
- 9-3. Spot drilling cycle (G81)
- 9-4. Counter boring cycle (G82)
- 9-5. Deep hole peck drilling cycle (G83)
- 9-6. High speed peck drilling cycle (G73)
- 9-7. Right-hand tapping cycle (G84)
- 9-8. Left-hand tapping cycle (G74)
- 9-9. Boring cycles (G85, G86)
- 9-10. Back boring cycle (G87)
- 9-11. Boring cycle with manual retraction (G88)
- 9-12. Boring cycle with dwell (G89)
- 9-13. Fine boring cycle (G76)
- 9-14. Polar coordinate system
- 9-15. Local coordinate system using datum shift (G52)
- Review questions 9
- Exercise 9

Chapter 10 SUBPROGRAMMING

- 10-1. Writing a subprogram
- 10-2. Multiple operations on specific set of hole locations
- 10-3. Pattern repeating
- 10-4. Coordinate system mirroring
- 10-5. Coordinate system rotation
- 10-6. Scaling
- Review questions 10
- Exercise 10

Chapter 11 MACRO PROGRAMMING

- 11-1. Applications of macros
 - (1) Part family
 - (2) Development of canned cycles
 - (3) Complex interpolations
 - (4) Controlling optional devices
- 11-2. Variables
 - (1) As arguments
 - (2) In calculations
 - (1) Local variables [#1 – #33]
 - (2) Common variables [#100 – #149] [#500 – #532]
 - (3) System variables [#1000 onwards]
- 11-3. Arithmetic expressions
- 11-4. Flow control statements
- 11-4-1. Unconditional branching



CNC FUNDAMENTALS AND PROGRAMMING
DETAILED CONTENTS

- 11-4-2. Conditional branching
- 11-4-3. Looping
- 11-5. Calling a macro
- Review questions 11
- Exercise 11

Chapter 12 DNC AND DATA COMMUNICATION

- 12-1. History
- 12-2. Direct numerical control
- 12-3. Distributed numerical control
- 12-4. DNC software features
 - (1) File transfer
 - (2) File management
 - (3) NC editor
 - (4) Report generation
 - (5) Access control
- 12-5. Advantages of DNC
- 12-6. DNC communications
 - 12-6-1. Parallel communications
 - 12-6-2. Serial communications
 - (1) Synchronous serial communication
 - (2) Asynchronous serial communication
- 12-7. Networking
 - 12-7-1. Networking components
 - (1) Coaxial cables
 - (2) Twisted pair cables
 - (3) Optical fiber cables
 - 12-7-2. Network topologies
 - (1) Physical topology
 - (2) Logical topology
 - (1) Bus network topology
 - (2) Ring network topology
 - (3) Star network topology
 - (4) Mesh network topology
 - (5) Tree network topology
- 12-8. Networking of CNC machine tools
 - (1) Wired DNC Networks
 - (i) RS232 DNC system
 - (ii) Ethernet DNC system
 - (2) Wireless DNC networks
 - (i) Ethernet wireless DNC system
 - (ii) Bluetooth wireless DNC system
- Review questions 12

Chapter 13 APT PROGRAMMING

- 13-1. Introduction
 - (1) Arithmetic calculations
 - (2) Cutter offset computation
 - (3) Postprocessing
- 13-2. Coordinate system
- 13-3. Language features
 - 13-3-1. Keywords
 - (1) Major words
 - (2) Minor words
 - 13-3-2. Symbols
 - Subscripted symbols
 - 13-3-3. Other features
 - (1) Numbers
 - (2) Angles
 - (3) Arithmetic operators
 - (4) Library functions
- 13-4. APT statements
 - 13-4-1. Postprocessor statements
 - 13-4-2. Auxiliary statements
 - 13-4-3. Geometry statements
 - (1) Point
 - (2) Line
 - (3) Circle
 - (4) Plane
 - (5) Pattern
 - (6) Vector
 - (7) Zsurf
 - 13-4-4. Motion statements
 - (1) Tolerance specifications
 - (2) Startup and positioning statements

- (3) Part, drive and check surfaces
- (4) Startup statements
- (5) Point to point motion statements
- 13-5. Machining center canned cycles
- 13-6. Macro programming
- 13-7. Repetitive programming
 - (1) Tracut
 - (2) Looping and branching
- Review questions 13
- Exercise 13

Chapter 14 CAD/CAM INTEGRATION

- 14-1. CAD/CAM systems
 - (1) Standalone CAD/CAM systems
 - (2) Integrated CAD/CAM/CAE systems
 - (1) Input of geometrical and technological data
 - (2) Calculation of the tool path and creation of the tool paths
 - (3) Translation of the tool paths to a part program
- Step 1: Create the manufacturing model
- Step 2: Setup the manufacturing data base and operations
 - (1) Tooling data
 - (2) Machine tools/work cells
 - (3) Fixture setups
 - (4) Operation setup
- Step 3: Define the NC-sequences
 - (1) Defining the machining features
 - (2) Define the sequence of machining features
- Step 4: Create the toolpath (cutter location files)
- Step 5: Post-processing
- 14-2. Evaluation of CAM software
 - 14-2-1. Turning
 - (1) 2-axis turning
 - (i) Contouring
 - (ii) Hole making
 - (iii) Roughing
 - (iv) Copy roughing
 - (v) Grooving
 - (vi) Threading
 - (2) Multi-spindle and multi-turret turning
 - 14-2-2. Milling
 - (1) Simple milling
 - (i) Pocketing
 - (ii) Facing
 - (iii) Hole making
 - (iv) Profiling
 - (v) Thread milling
 - (2) Advanced milling
 - (i) Rotary axis
 - (ii) Machining using 4th and 5th axis
 - (3) Machining of 3D freeform shapes
- 14-3. Conversational programming
 - (1) Initialization module
 - (2) Machining cycle module
 - (3) Contour module
 - (4) Tool module
 - (5) Toolpath simulation module
 - (6) Utility module
- 14-4. Part program verification
 - 14-4-1. The purposes of part program verification
 - (1) To detect geometric errors of the cutter path
 - (2) To detect potential tool interference
 - (3) To detect incorrect cutting conditions
 - 14-4-2. Part program verification methods
 - (1) Dry run
 - (2) Machine a prototype
 - (3) CAD-based verification software
- Review questions 14

Appendix I MULTIPLE CHOICE QUESTIONS

Appendix II NC/CNC MANUAL PART PROGRAMMING CODES

INDEX

