

Sorin HERLE

**DIGITAL CONTROL OF
MACHINE TOOLS**



**UTPRESS
Cluj-Napoca, 2017
x ISBN**



**DIGITAL CONTROL
OF MACHINE TOOLS
- PRACTICAL EXERCISES -**

*"For the things we have to learn before we can do them, we learn
by doing them."*

(Aristotle)

CNC machines are automatic devices computer controlled, capable of carrying out a series of machining operations with very high accuracy.

In modern manufacturing systems, with a high degree of flexibility, the CNC machines became virtually indispensable.

Originally, the numerical control has been used mainly to automate lathes and milling machines. Today, computer numerical control is used for many types of equipment such as robots, laser cutting machines, water jet cutting machines, plasma cutting machines, 3D printers, etc. However, most often, computer numerical control is used for milling and turning machines.

This volume is addressed primarily to students enrolled in the Automation and Applied Informatics study program, but it can be useful to anyone interested in the field of computer numerical control.

This volume contains 14 units that gradually make the transition from the programming basics of CNC machines, to more advanced concepts related to design and automatic generation of CNC programs using CAD-CAM systems.

Each unit presents some theoretical aspects accompanied by practical examples explained step by step. To deepen the knowledge presented, some exercises are proposed at the end of each unit.

I am grateful to fourth year students, Automation and Applied Informatics class of 2016, who studied during a semester the units included in this book and came up with very useful comments which have led to an improved content. Especially, I would like to thank to the following students: Adrian Pop, Alexandra Mânzat, Călin Cîmpan, Cătălin Cîmpan, Cosmin Țurcaș, Daniela Pop, Diana Baci, Gabriel Hossu, Horațiu Vilt, Larisa Ember, Levi Ciupe, Nela Suciu, Simida Rodilă, Vasile Boancă. I also thank Catalin Tămaș for designing the cover of this book.

I hope this book will be useful to anyone interested in programming CNC machines. Suggestions and comments for improving this work are welcome.

Sorin HERLE





CONTENTS



UNIT 1: THE EXSL-WIN FOR PROGRAMMING OF CNC MACHINES 9

PART I: EXSL-WIN FOR MILLING MACHINES

What is EXSL-WIN? 12

How to create a project for a CNC milling machine? 13

What are the preliminary steps for writing a program? 14

How to define the raw material? 14

How to choose the workpiece zero point? 15

How to configure the magazine? 16

How to write a program? 18

How to run a program? 19

How to change the view of the workpiece? 19

How can be seen the position of the tool? 20

Example 1 21

PART II: EXSL-WIN FOR TURNING MACHINES

What is EXSL-WIN? 23

How to create a project for a CNC turning machine? 24

What are the preliminary steps for writing a program? 25

How to define the raw material? 25

How to choose the workpiece zero point? 26

How to configure the turret? 27

How to write a program? 29

How to run a program? 30

How to change the view of the workpiece? 30

How can be seen the position of the tool? 31

Example 2 32

Test 34

Score 34

UNIT 2: PROGRAMMING LINEAR MOTIONS ON A CNC MILLING MACHINE 35

What is the linear motion? 36

What types of linear motions can be programmed on a CNC milling machine? 36

What is the difference between absolute and incremental coordinates? 37

What command is used, for a CNC milling machine, to program a linear motion? ... 37

What is the feed rate (F)? 38

What is rapid positioning? 39

Example 1 39

Example 2 41

Example 3 43

Application 1 44

Application 2 45

Test 47

Score 47

UNIT 3: PROGRAMMING CIRCULAR MOTIONS ON A CNC MILLING MACHINE 49

What is the circular motion? 50

What is a circle? 50

What types of circular motions can be programmed on a CNC milling machine? 51

<i>What are the commands used for programming circular motions for a CNC milling machine?</i>	53
<i>How to choose the sign for the parameters I, J, K?</i>	54
<i>Example 1</i>	57
<i>Example 2</i>	59
<i>Application 1</i>	61
<i>Application 2</i>	62
<i>Test</i>	65
<i>Score</i>	65
UNIT 4: PROGRAMMING FACE MILLING AND SIDE MILLING OPERATIONS	67
<i>What is face milling?</i>	68
<i>How is the face milling operation performed?</i>	69
<i>What types of trajectory can be used for face milling?</i>	70
<i>How is the number of passes influenced by the width of cut and the tool diameter?</i>	70
<i>Example 1</i>	70
<i>What is the compensation for the cutter radius?</i>	73
<i>Example 2</i>	74
<i>Application</i>	76
<i>Test</i>	79
<i>Score</i>	79
UNIT 5: PROGRAMMING THE POCKET MILLING OPERATIONS	81
<i>What is a pocket?</i>	82
<i>What cutting tools are used for machining pockets?</i>	83
<i>What are the factors that must be considered for programming pockets?</i>	83
<i>How to program the cutting tool entry?</i>	85
<i>What types of trajectories can be used for machining a rectangular pocket?</i>	85
<i>What types of trajectories can be used for machining a circular pocket?</i>	86
<i>What are arc-in and arc-out motions?</i>	87
<i>Example 1</i>	89
<i>Example 2</i>	91
<i>Application</i>	92
<i>Test</i>	95
<i>Score</i>	95
UNIT 6: PROGRAMMING DRILLING OPERATIONS ON A CNC MILLING MACHINE	97
<i>What are the characteristics of a hole?</i>	98
<i>What are the most common operations for machining a hole?</i>	98
<i>What types of tools are used for machining holes on a CNC milling machine?</i>	99
<i>What operations are necessary for drilling a hole?</i>	100
<i>What methods are available for drilling a hole on a CNC milling machine?</i>	100
<i>What calculations are required for spot drilling operations?</i>	102
<i>Example 1</i>	102
<i>What calculations are required for machining blind holes?</i>	103
<i>Example 2</i>	104
<i>What calculations are required for machining through holes?</i>	105

Example 3 106
What calculations are required for countersinking? 107
Example 4 108
How to thread a hole? 109
What is a subprogram? 109
Application 110
Test 113
Score 113

UNIT 7: PROGRAMMING THE MILLING CYCLES 115

What are the cycles? 116
What are the advantages and disadvantages of the cycles? 116
For what types of milling operations it is possible to use cycles? 116
How to program CYCLE 71? 117
Example 1 119
How to program CYCLE 72? 120
Example 2 121
How to program the cycle POCKET1? 123
Example 3 124
How to program the cycle POCKET2? 125
Example 4 126
How to program CYCLE90? 127
Example 5 127
Example 6 129
Application 130
Test 134
Score 134

UNIT 8: PROGRAMMING THE MILLING AND DRILLING CYCLES 135

What are the cycles? 136
What are the advantages and disadvantages of the cycles? 136
For what types of milling and drilling operations it is possible to use cycles? 136
How to program the cycle SLOT1? 137
Example 1 138
How to program the cycle SLOT2? 139
Example 2 141
How to program CYCLE83? 142
Example 3 143
How to program the cycle HOLES1? 144
Example 4 145
How to program the cycle HOLES2? 146
Example 5 147
Application 148
Test 151
Score 151

UNIT 9: PROGRAMMING OF THE TURNING OPERATIONS 153

<i>What is turning?</i>	154
<i>What are the most common turning operations?</i>	154
<i>What tools are used for turning operations?</i>	154
<i>What is face turning?</i>	155
<i>Example 1</i>	156
<i>What is plain turning or straight turning?</i>	157
<i>Example 2</i>	157
<i>What is contouring?</i>	159
<i>Example 3</i>	159
<i>What is drilling?</i>	161
<i>Example 4</i>	161
<i>What is threading?</i>	163
<i>What are the commands used for threading?</i>	163
<i>Example 5</i>	164
<i>What is grooving?</i>	166
<i>Example 6</i>	166
<i>What is chamfering?</i>	167
<i>Application</i>	168
<i>Test</i>	170
<i>Score</i>	170
UNIT 10: PROGRAMMING THE TURNING CYCLES	171
<i>What are the turning cycles?</i>	172
<i>What are the advantages and disadvantages of the cycles?</i>	172
<i>For what types of turning operations it is possible to use cycles?</i>	172
<i>How to program the cycle L93?</i>	173
<i>Example 1</i>	174
<i>Example 2</i>	176
<i>How to program the cycle L95?</i>	177
<i>Example 3</i>	179
<i>How to program the cycle L98?</i>	180
<i>Example 4</i>	182
<i>How to program the cycle L97?</i>	184
<i>Example 5</i>	185
<i>Application</i>	188
<i>Test</i>	191
<i>Score</i>	191
UNIT 11: PROGRAMMING THE MILLING OPERATIONS IN CAPSMILL	193
<i>What is CADEM CAPSmill?</i>	194
<i>How to visualize the machining operations and the automatically generated program code?</i>	195
<i>Example 1</i>	195
<i>How to create a new part?</i>	196
<i>How to define the setup?</i>	197
<i>How to make the drawing of the workpiece?</i>	198
<i>How to define the blank?</i>	204

How to define the machining operations? 204
How to choose the controller? 216
How to simulate the machining operations? 216
How to generate and visualize the NC program? 217
Application 218
Test 222
Score 222

UNIT 12: PROGRAMMING THE TURNING OPERATIONS IN CAPSTURN 223

What is CADEM CAPSturn? 224
How to visualize the machining operations and the automatically generated program code? 225
Example 1 225
How to create a new part? 226
How to define the setup? 227
How to make the drawing of the workpiece? 228
How to define the blank? 233
How to define the machining operations? 236
How to choose the controller? 243
How to simulate the machining operations? 243
How to generate and visualize the NC program? 244
Application 245
Test 248
Score 248

UNIT 13: PROGRAMMING THE PC F020 MILLING MACHINE 249

What is EXSL-WIN? 251
What are the characteristics of the CNC milling machine? 252
What are the steps for machining a workpiece? 253
How to power on the system? 254
How to open EXSL-WIN? 255
How to create a new project? 255
How to define the blank? 256
How to choose the workpiece zero point? 258
How to configure the magazine? 259
How to edit a program? 261
How to run a program in simulator? 263
How to connect to the real CNC machine? 264
How to initialize the CNC machine? 265
How to secure the blank in the vise? 266
How to mount the edge finder into the spindle? 266
How to find the workpiece zero point? 267
How to add tools in magazine? 269
How to dry run a program? 269
How to modify a program? 270
How to run a program step by step? 270
How to run the program continuously? 271

<i>What is analyzing the finished part?</i>	271
<i>Application</i>	272
<i>Test</i>	274
<i>Score</i>	274
UNIT 14: PROGRAMMING THE CH-A01 MILLING MACHINE	275
<i>What are the characteristics of the CNC milling machine?</i>	277
<i>How to control the milling machine?</i>	278
<i>What are the steps for machining a workpiece?</i>	280
<i>How to power on the system?</i>	280
<i>How to open Universal Gcode Sender?</i>	281
<i>How to establish the communication?</i>	281
<i>How to secure the blank in the vise?</i>	282
<i>How to find the workpiece zero point?</i>	282
<i>How to edit a program?</i>	283
<i>How to simulate a program?</i>	283
<i>How to dry run a program?</i>	284
<i>How to modify a program?</i>	284
<i>How to run continuously a program?</i>	284
<i>What is analyzing the finished part?</i>	285
<i>Application</i>	286
<i>Test</i>	288
<i>Score</i>	288
BIBLIOGRAPHY	289
ANNEX 1: Commands for Siemens Sinumerik 840D (milling machine)	291
ANNEX 2: Commands for Siemens Sinumerik 810T (turning machine)	293
ANNEX 3: Dictionary of common terms	295

CODE	DESCRIPTION
G00	Rapid traverse
G01	Linear movement
G02	Clockwise circular movement.
G03	Counterclockwise circular movement
G04	Dwell
G17	Select X-Y plane.
G18	Select Z-X plane.
G19	Select Y-Z plane.
G20	Program coordinates are inches
G21	Program coordinates are mm
G40	Cancel tool radius compensation
G41	Left tool radius compensation
G42	Right tool radius compensation
G54	Work coordinates system
G55	Work coordinates system
G56	Work coordinates system
G57	Work coordinates system
G90	Absolute coordinates
G91	Incremental coordinates
M0	Program stop
M1	Optional stop
M2	End of program
M3	Spindle rotates clockwise

M4	Spindle rotates counterclockwise
M5	Spindle stop.
M6	Tool changer
M7	Mist coolant ON
M8	Flood coolant ON
M9	Coolant OFF
M17	End of subprogram
M30	End of program
F	Feed rate
S	Spindle speed
T	Tool
D	Tool correction
CYCLE71	Face milling cycle
CYCLE 72	Path milling cycle
POCKET1	Rectangular pocket milling cycle
POCKET2	Circular pocket milling cycle
CYCLE90	Thread cutting cycle.
SLOT 1	Slot on a circle cycle
SLOT 2	Circumferential slot cycle
CYCLE83	Stepwise drilling cycle
HOLE1	Row of holes cycle
HOLE2	Hole circle cycle

CODE	DESCRIPTION
G00	Rapid traverse
G01	Linear movement
G02	Clockwise circular movement.
G03	Counterclockwise circular movement
G20	Program coordinates are inches
G21	Program coordinates are mm
G33	Thread cutting with constant lead
G34	Thread cutting with linear increasing lead
G35	Thread cutting with linear decreasing lead
G40	Cancel tool nose radius compensation
G41	Left tool nose radius compensation
G42	Right tool nose radius compensation
G54	Work coordinates system
G55	Work coordinates system
G56	Work coordinates system
G57	Work coordinates system
G90	Absolute coordinates
G91	Incremental coordinates
G94	Feed per minute
G95	Feed per revolution
G96	Constant cutting speed, feed per revolution
G97	Constant spindle speed
M0	Program stop

M1	Optional stop
M2	End of program
M3	Spindle rotates clockwise
M4	Spindle rotates counterclockwise
M5	Spindle stop.
M6	Tool changer
M7	Mist coolant ON
M8	Flood coolant ON
M9	Coolant OFF
M17	End of subprogram
M30	End of program
F	Feed rate
S	Spindle speed
T	Tool
D	Tool correction
L	Prefix of the name of a subprogram
L93	Grooving cycle
L95	Paraxial turning cycle
L97	Threading cycle
L98	Deep-hole drilling cycle

EXPRESION	DESCRIPTION
Absolute coordinates	A series of numerical positions that are calculated from a fixed point of origin.
Approach distance	The distance between the lateral edge of the workpiece and lateral edge of the tool, before the tool enter into the workpiece or after the tool exit from the workpiece.
Blind hole	It is a hole that does not penetrate the workpiece
Boring	The process of enlarging a hole that has already been drilled.
Chamfering	Machining an angled edge around the end of a cylindrical workpiece.
Clearance	Any useful space that is intentionally maintained between components.
Contouring	It is a turning operation along the contour of a cylindrical workpiece.
Countersinking	It is an operation that enlarges an existing hole in a conical shape, to a required depth.
Cycle	Special command that allows programming of certain machining operations in a single line of program (rarely two lines for certain types of controllers).
Deep drilling	It involves a stepwise motion with return to a fixed point for chips removal.
Drill	A machining tool used to penetrate the surface of a workpiece and make a round hole.
Drilling	It involves a continuous motion in the longitudinal direction of the hole.
Face turning	The face turning operation is performed by moving the cutting tool on the transverse direction of the workpiece.
Feed rate	The rate of tool travel through the workpiece per unit of time, generally expressed in inch or mm per revolution (turning) or inch or mm per tooth.
Helical motion	It is a circular motion in a plane (X-Y, X-Z, Y-Z), combined with a linear motion in the direction perpendicular to the plane in which the circular motion take place.
Incremental coordinates	A series of numerical positions that use the previous position as the point of origin for the next position.
MCALL	It is used for modal calling of a cycle.
Modal command	The command that remains in a certain mode until it is canceled by an equivalent mode.
Non-modal command	The non-modal command affects only the line on which it occurs.
Peck drilling	It involves a stepwise motion with return to a variable point for chips breaking.
Plain turning	The plain turning operation produces a cylindrical surface by removing excess material from the blank, while the tool moves along the blank.
Pocket	An interior recess that is cut into the surface of a workpiece. Pockets may be round or rectangular.

Preparatory G codes	Preparatory codes identify the type of activities the machine will execute.
Spot drill	A short, sturdy drill used to start a hole and accurately locate it. The spot drills have a 90° or 60° tips.
Tap	Tool used for machining internal threads
Threading	Threading is the process of creating a screw thread.
Through hole	It is a hole that penetrates the workpiece
Tool entry/exit	It refers to the trajectory followed by the cutting tool to get in contact with the workpiece.
Tool nose radius compensation	An offset feature used on a turning center that slightly shifts the toolpath for the rounded tip of an insert during contouring, chamfering, and other multi-axis operations.
Tool radius compensation	It adjusts the position of the tool center so that its edge to be in contact with the workpiece.
Turning	The machining process used to make cylindrical parts.
Width of cut (step-over)	Width of cut or step-over is the contact area between the tool and the workpiece during machining. It is usually expressed as a percentage of the tool diameter.

In modern manufacturing systems, with a high degree of flexibility, the CNC machines became virtually indispensable.

Originally, the numerical control has been used mainly to automate lathes and milling machines.

Today, computer numerical control is used for many types of equipment such as robots, laser cutting machines, water jet cutting machines, plasma cutting machines, 3D printers, etc. However, most often, computer numerical control is used for milling and turning machines.

This volume contains 14 units that gradually make the transition from the programming basics of CNC machines, to more advanced concepts related to design and automatic generation of CNC programs using CAD-CAM systems.

Each unit presents some theoretical aspects accompanied by practical examples explained step by step. To deepen the knowledge presented, some exercises are proposed at the end of each unit.