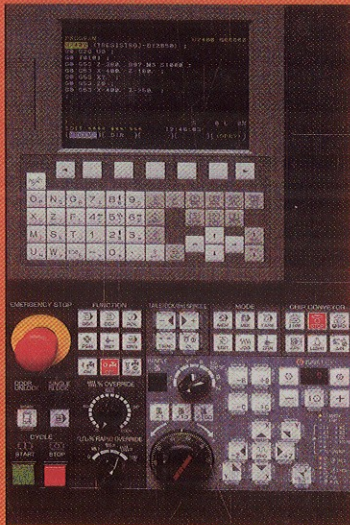




CNC **CONTROL** **SETUP** **for** **Milling and Turning**

MASTERING CNC CONTROL SYSTEMS

Peter Smid

Table of Contents

Chapter 1		
CONCEPTS OF CNC MACHINING		1
CNC PROCESS		1
Drawing Evaluation		1
Material Identification		1
Part Holding		2
Tooling Selection		2
Cutting Conditions		2
Program Writing		2
Program Verification		2
Program Documentation		3
Program Transfer		3
WORK COMPLETION		3
Program Evaluation		3
Material Check		3
Tooling Preparation		3
Tooling Setup		3
Fixture Setup		4
Program Loading		4
Setting Offsets		4
First Part Run		4
Program Optimization		4
Production Run		4
Part Inspection		4
Safety Issues		4
Chapter 2		
CNC MACHINE SPECIFICATIONS		5
MACHINE SPECIFICATIONS		5
Useful Information		5
VERTICAL MACHINING CENTER		5
Typical Specifications - VMC		6
HORIZONTAL MACHINING CENTER		6
Typical Specifications - HMC		6
LATHES AND TURNING CENTERS		7
Typical Specifications		7
FANUC SYSTEM SPECIFICATIONS		7
Controlled Axes		7
Operation Functions		8
Interpolation Functions		8
Feedrate Functions		8
Spindle Functions		8
Tool Functions		9
Program Input		9
Editing Functions		9
Setting and Display Functions		9

	<i>Data Input / Output</i>	10
	<i>Options</i>	10
Chapter 3		
PROGRAM INTERPRETATION		11
	PROGRAM STRUCTURE	11
	<i>Logical Structure</i>	11
	<i>Program Structure Consistency</i>	14
	PROGRAM INTERPRETATION	14
	<i>Units</i>	15
	<i>Type of Operation</i>	15
	<i>Speeds and Feeds</i>	15
	<i>Tool Numbers</i>	15
	<i>Dwell Time</i>	16
	<i>Other Considerations</i>	16
	PROGRAM ERRORS	17
	<i>Syntax Errors</i>	17
	<i>Logical Errors</i>	17
	<i>Offset Errors</i>	18
	<i>Incorrect Units</i>	18
Chapter 4		
CONTROL SYSTEM		19
	CONTROL SYSTEM OVERVIEW	19
	SYSTEM MEMORY	20
	<i>Paper Tape</i>	20
	<i>Memory Capacity</i>	21
	CONTROL PANEL	22
	<i>ON-OFF Buttons</i>	23
	<i>Display Screen and Soft Keys</i>	23
	<i>Selection Keys</i>	23
	<i>Address Keyboard</i>	25
	<i>Shift Key</i>	25
	<i>Numeric Keyboard</i>	25
	<i>Edit Keys</i>	26
	<i>Page and Cursor Keys</i>	26
	<i>EOB - CAN - INPUT Keys</i>	26
	<i>Reset Key</i>	26
	<i>Help Key</i>	26
Chapter 5		
OPERATION PANEL		27
	OPERATION PANEL LAYOUT	28
	MAIN BUTTONS AND SWITCHES	28
	<i>Power ON / Ready Light Indicator</i>	28
	<i>Cycle Start / Feedhold - Auto Operation</i>	28
	STATUS INDICATOR LIGHTS	29
	<i>Indicator Lights - Three Groups</i>	29
	ON/OFF SWITCHES	32
	<i>Toggle and Button Switches</i>	33

<i>Program Testing and Setup Mode</i>	37
ROTARY SWITCHES	37
<i>Mode Selection</i>	37
<i>Axis Selection</i>	41
OVERRIDES	41
<i>Rapid Override</i>	41
<i>Feedrate Override</i>	42
<i>Spindle Override</i>	44
<i>Jog Feedrate Selection</i>	44
EDIT KEY	45
EMERGENCY STOP SWITCH	45
OTHER FEATURES	45
CONTROL PANEL SYMBOLS	45
<i>Switches</i>	46
<i>Modes</i>	46
<i>Operations</i>	47
 Chapter 6	
SETUP HANDLE	49
PURPOSE	49
DESIGN	49
HANDLE FEATURES	50
<i>Mode Selection</i>	50
<i>Axis Selection</i>	50
<i>Range Selection</i>	50
<i>Dial Wheel</i>	50
APPLICATIONS	52
<i>Manual Absolute Switch</i>	52
<i>Safety Issues</i>	52
 Chapter 7	
MILLING TOOLS - SETUP	53
MILLING TOOLS	53
<i>Tool Holders</i>	53
<i>Cutting Tools</i>	54
<i>Tooling System</i>	54
ATC - AUTOMATIC TOOL CHANGE	55
<i>Manual Tool Change</i>	55
<i>Tool Magazine</i>	56
<i>Fixed Type ATC</i>	57
<i>Random Memory Type ATC</i>	57
REGISTERING TOOLS	58
TOOL SPECIFICATIONS	58
<i>Maximum Tool Length</i>	58
<i>Maximum Tool Diameter</i>	59
<i>Maximum Tool Weight</i>	59
ATC PROCESS	60
TOOL REPETITION	62
<i>Repeating the Same Tool</i>	62

Repeating a Different Tool	62
Solving the Problem	63
Tool Change - One or Two Blocks	64
DUMMY TOOL	64
SUMMARY	64
 Chapter 8	
SETTING PART ZERO	65
SYMBOLS	65
WHAT IS PART ZERO ?	65
Part Zero Location	65
VICES FOR CNC WORK	67
CNC Vise - Basic Design	67
CNC Vise - Setup	67
APPLICATION EXAMPLE	68
Part Orientation	68
Vise Orientation	69
FINDING A CENTER	71
PART ZERO ON CNC LATHES	71
X-axis Zero	71
Z-axis Zero	71
Work Shift	72
 Chapter 9	
WORK OFFSET SETTINGS	73
REFERENCE POINTS	73
Machine Zero	73
Part Zero	73
Machine Zero to Part Zero Connection	73
PART SETUP EVALUATION	73
USING WORK OFFSETS	74
Preparatory Commands G54-G59	75
Establishing the Connection	75
USING EDGE FINDERS	77
Magnetic Single Edge Finder	77
Magnetic Center Finder	77
DISTANCE-TO-GO	78
SHAFT TYPE EDGE FINDERS	78
Prerequisites	78
Individual Steps	79
Work Offset Calculations	79
Original Example Revisited	81
WORK OFFSET Z-SETTING	81
Where is the Z-setting?	81
Setting the Z-axis	82
MULTIPLE WORK OFFSETS	82
Program Entry	83
WORK OFFSET ADJUSTMENTS	83
Mixing Units of Measurement	83

<i>Height of Part</i>	84
EXT WORK OFFSET	85
<i>EXT or COM ?</i>	85
<i>Offset Adjustment Options</i>	85
WORK OFFSETS - DATA INPUT	86
<i>Data Change Example</i>	86
 Chapter 10	
TOOL LENGTH OFFSET	87
TOOL LENGTH	87
<i>Tool Length Offset Register</i>	87
TOOL LENGTH OFFSET COMMANDS	87
<i>Limitations of Work Offsets</i>	88
<i>G43 and G44 Commands</i>	88
<i>Applying Tool Length Offset</i>	89
<i>G49 Command</i>	89
OFFSET MEMORY TYPE	89
<i>Offset Memory - Type A</i>	90
<i>Offset Memory - Type B</i>	90
<i>Offset Memory - Type C</i>	90
<i>INPUT and +INPUT Key Selection</i>	90
<i>Adjustment Examples</i>	91
MACHINE GEOMETRY	92
SETTING THE TOOL LENGTH	92
<i>Tool Assembly Fixture</i>	93
<i>Preset Tool Method</i>	93
<i>Touch-Off Method</i>	95
<i>Reference Tool Method</i>	97
EXT OFFSET AND WORK OFFSET	100
WORK AND TOOL LENGTH OFFSET	100
 Chapter 11	
MACHINING A PART	101
MACHINING OBJECTIVES	101
PROCESS OF MACHINING	101
DRAWING EVALUATION	101
<i>Tolerances</i>	101
<i>Radius from I and J</i>	102
<i>Surface Finish</i>	103
<i>Stock Allowance</i>	103
MATERIAL INSPECTION	103
PROGRAM EVALUATION	103
<i>Programs Generated Manually</i>	104
<i>Programs Generated by Software</i>	106
<i>Part Setup and Tooling</i>	106
PROGRAM INPUT	106
<i>Memory Mode</i>	106
<i>DNC Mode</i>	106
PROGRAM VERIFICATION	106

Manual Verification	107
Software Verification	107
PRODUCTION MACHINING	107
Responsibilities	107
PART INSPECTION	108
PROGRAM SOURCES	108
CAM Programs and Machining	108
Conversational and Macro Programming	108
PREVENTING A SCRAP	109
Causes of Scrap	109
Steps to Eliminate Scrap	109
Let Offsets Work for You	109
Offset Adjustment - Example	110
PROGRAM MODIFICATION	111
MAINTAINING SURFACE FINISH	111
Surface Finish in a Drawing	111
Effect of Cutting Tools	111
Selection of Cutting Data	112
Machining Direction	112
Chatter / Vibration	113
COOLANTS	113
Benefits of Coolants	114
Working with Coolant	114
 Chapter 12	
MACHINING HOLES	115
FIXED CYCLES - OVERVIEW	115
DRILLING HOLES	115
Drilling Example	116
Terminology	116
Initial Level and R-Level	117
Depth	117
Spot Drilling + Counterbore + Countersink	117
Peck Drilling	118
Changing Q-depth	119
TAPPING OPERATIONS	120
Tap Drill	120
Feedrate Calculation	120
Troubleshooting	121
Rigid Tapping	122
BORING OPERATIONS	123
G85 Cycle	123
G86 Cycle	123
G87 Cycle	123
G88 Cycle	123
G89 Cycle	124
G76 Cycle	124
DRILLING FORMULAS	125
Center Drills	125
Spot Drills	126

Twist Drills	126
Chapter 13	
MULTIPART SETUP	127
MULTIPLE PARTS	127
Setup Conditions	127
ORIGIN DISTANCES UNKNOWN	127
Program Data	128
Setup Process	129
ORIGIN DISTANCES KNOWN	129
Program Data	130
Using Work Offsets	130
Using Local Coordinate System	130
Using G10 Setting	130
Using Custom Macro	131
EXTENDED WORK OFFSETS	132
Available Range	132
Standard vs. Extended Work Offset Range	132
Z-AXIS AND MULTIPART SETUP	132
Variable Height	132
Tool Length Offsets	132
PROGRESSIVE SETUP	134
SUMMARY	134
Chapter 14	
CUTTER RADIUS OFFSET	135
BENEFITS	135
PROGRAM DATA	135
Address D	135
Offset Memory Types	135
Equidistant Toolpath	136
Cutting Direction	136
WORKING WITH RADIUS OFFSET	137
D-offset Number	137
Look-Ahead Type Offset	138
INITIAL CONSIDERATIONS	139
Cutter Diameter	139
Effects of Machining	140
Scrap Prevention	140
ACTUAL OFFSET SETTING	140
Programmed Tool Size	141
External Offset Settings	141
Internal Offset Settings	142
Conclusion	142
USING OFFSET MEMORY TYPES	142
Geometry and Wear Offsets	142
Offset Memory - Type A	142
Offset Memory - Type B	143
Offset Memory - Type C	144

PROGRAMS FROM COMPUTER	144
<i>Mastercam Example</i>	144
SOLVING PROBLEMS	145
<i>Alarm Number 041</i>	145
<i>Insufficient Clearance</i>	146
<i>Cutter Radius Too Large</i>	147
<i>Contour Start / End Error</i>	148
CHANGE of TOOL DIAMETER	149
<i>Units Conversion</i>	150
 Chapter 15	
TOOLS FOR CNC LATHES	151
PROGRAMMMING LATHE TOOLS	151
<i>Zero Wear Offset</i>	151
<i>Feedrate Error</i>	151
<i>Prevention</i>	152
TOOL NUMBERS	152
TOOL TURRET	152
TOOL HOLDERS	153
<i>Standards</i>	153
<i>Identification</i>	153
<i>ISO and ANSI</i>	154
<i>Gage Insert</i>	154
<i>Other Tool Holders</i>	154
CUTTING INSERTS	156
<i>Insert Shape Identification</i>	156
<i>ISO Unique Identification</i>	157
<i>ANSI Unique Identification</i>	158
<i>Grades, Coating and Chip Breakers</i>	158
CHANGING AN INSERT	158
<i>Corner Radius Change</i>	158
TOOL CHANGING	159
LATHE CHUCK	160
<i>Chuck Clamp and Unclamp</i>	160
<i>Chuck Pressure</i>	160
SELECTION OF JAWS	160
<i>Hard Jaws</i>	160
<i>Soft Jaws</i>	160
 Chapter 16	
LATHE OFFSETS	161
Lathe Geometry	161
<i>Machine Zero</i>	161
<i>Part Zero</i>	162
<i>Tool Reference Point</i>	162
WORK SETUP	163
<i>Definitions</i>	163
<i>Geometry Offset</i>	163
<i>The R and T Columns</i>	164

Wear Offset	165
DRAWING AND PROGRAM	165
Changing Tool Numbers	166
Bar Extension	166
MANAGING TOLERANCES	166
Tolerances and Program Data	167
Tolerances on Diameters - Part A	167
Tolerances on Shoulders - Part B	169
Tolerances on Diameters and Shoulders	170
WORKING WITH WEAR OFFSETS	171
Common Offset Change Example	171
WEAR OFFSET PARAMETERS	172
 Chapter 17	
TURNING AND BORING	173
LATHE CYCLES G70 - G71 - G72	173
Concepts of Turning and Boring Cycles	174
Program Listing	175
Contour Shape - External	176
G71 Cycle Format for Two Blocks	177
Contour Shape - Internal	177
PROGRAM FEATURES TO WATCH	178
Start Point Location	178
Clearances	179
Tool Change Clearance	180
Program after Changes	181
PROGRAM DATA TO WATCH	182
Start Point Too High or Too Low	182
Positive and Negative Stock Allowance	182
W-Stock Amount Too Heavy	182
Return to Start Point in the Cycle	183
P and/or Q Reference Defined Incorrectly	183
Feedrates for Roughing and Finishing	184
Cutter Radius Offset	184
Small Inside Radius	184
G72 AND G73 TURNING CYCLES	185
G72 Lathe Cycle	185
G72 Cycle Format for Two Blocks	185
G73 Pattern Repeating Cycle	186
G73 Cycle Format For Two Blocks	186
ONE-BLOCK FORMAT CYCLES	186
G71 Cycle Format for One Block	186
G72 Cycle Format for One Block	187
G73 Cycle Format for One Block	187
G71 TYPE I AND TYPE II CYCLES	187
Type I	187
Type II	187
SUMMARY	188

Chapter 18

TOOL NOSE RADIUS OFFSET	189
BRIEF REVIEW	189
TOOLPATH EVALUATION	189
Program Listing - No Offset	190
Program Listing - With Offset	191
Offset Screen	191
FACING TO CENTERLINE	192
Tool Height Error	192
Incorrect Tool Position	192
G41 - Yes or No?	193
OFFSET CHANGE	193
CLEARANCE AMOUNT	194

Chapter 19

GROOVING ON LATHES	195
GROOVE MACHINING	195
The Simplest Groove	195
Standard Grooves	195
Typical Groove	196
Grooving Tool	196
Grooving Insert	196
Control Settings	197
Program Evaluation	197
Basic Groove Dimensions	198
TOLERANCES NOT SPECIFIED	198
Dimensional Problems	199
DEPTH CONTROL	199
External and Internal Grooves	200
POSITION CONTROL	201
Left Side Reference Point	201
Right Side Reference Point	201
Reference Point Set in the Middle	201
Offset Adjustment	201
WIDTH CONTROL	203
Program Requirements	203
POSITION AND WIDTH CONTROL	204
GEOMETRY OFFSET	204
Change of Tool Width	204
Program Requirements - Summary	205
GROOVING CYCLES	205
G74 Cycle	205
G75 Cycle	205

Chapter 20

THREAD CUTTING	207
THREADING PROCESS	207
THREADING TOOLS	207
Threading Tool Holders and Inserts	207

<i>Hand of Tool Selection</i>	208
THREADING INFEEED	210
<i>Radial Infeed</i>	210
<i>Flank Infeed</i>	210
<i>Modified Flank Infeed</i>	210
<i>Alternating Flank Infeed</i>	211
THREADING INSERT	211
<i>Full Profile</i>	211
<i>Partial Profile</i>	211
<i>Clearances</i>	212
<i>Multitooth Inserts</i>	212
THREAD RELATED DEFINITIONS	212
<i>Pitch</i>	212
<i>Lead</i>	212
<i>Other Definitions</i>	212
METHODS OF PROGRAMMING	213
<i>Longhand Threading - G32</i>	213
<i>Box Threading - G92</i>	213
<i>Repetitive Cycle Threading - G76</i>	214
THREADING DATA	214
<i>Depth of Thread</i>	214
<i>Depth of the First Pass</i>	214
<i>Threading Feedrate</i>	214
<i>Angle of Thread</i>	214
<i>Spring Pass</i>	214
G76 THREAD CUTTING CYCLE	215
<i>One-Block Format</i>	215
<i>Two-Block Format</i>	215
<i>G76 Cycle Examples</i>	216
MULTI START THREADS	218
SPINDLE SPEED LIMITATION	218
<i>Machine Specifications</i>	218
THREADING DEFAULTS	218
 Chapter 21	
WORKING WITH BLOCK SKIP	219
BLOCK SKIP FUNCTION	219
<i>Block Skip Switch</i>	219
<i>Using Block Skip</i>	220
<i>Block Skip within a Block</i>	220
<i>Variable Stock</i>	220
TRIAL CUTS	221
<i>Maintaining Tolerances</i>	221
<i>Trial Cut Concepts</i>	221
<i>Trial Cut Methods</i>	221
MILLING APPLICATIONS	222
<i>Program Sample</i>	222
<i>Program Evaluation</i>	222
<i>Finishing Tool - Program Details</i>	223
<i>Conclusion</i>	223

TURNING APPLICATIONS	223
<i>Test Cut Addition</i>	224
Chapter 22	
OFFSET CHANGE BY PROGRAM	225
LOCAL COORDINATE SYSTEM	225
<i>Work Offset Change</i>	226
DATUM SHIFT	226
<i>G10 Format - Milling Applications</i>	226
<i>G10 For Roughing And Finishing</i>	229
<i>G10 Format - Lathe Applications</i>	230
SETTING PARAMETERS BY G10	232
Chapter 23	
SYSTEM PARAMETERS	233
WHAT ARE PARAMETERS?	233
<i>Parameters for Operators</i>	233
<i>Fanuc Parameter Classification</i>	233
<i>Access to Parameters</i>	233
<i>Backing Up Parameters</i>	235
PARAMETER TYPES	235
<i>Bit Type Description</i>	236
<i>Practical Interpretation</i>	236
INPUT OF REAL NUMBERS	238
<i>Increment System</i>	239
<i>Units of Input</i>	239
<i>Data Range</i>	239
VALID DATA RANGES	240
PRACTICAL APPLICATIONS	240
PARAMETERS OF SETTING	240
<i>Setting (Handy)</i>	240
PARAMETERS OF FEEDRATE	241
PARAMETERS OF DISPLAY	243
<i>Programs Registration and Editing</i>	243
<i>Password Protection</i>	243
<i>Program Display</i>	244
<i>Automatic Block Numbering</i>	244
<i>Other Parameters</i>	244
PARAMETERS OF PROGRAMS	245
<i>Decimal Point</i>	245
<i>Power-On Defaults</i>	245
<i>End of Program Functions</i>	245
<i>Number of M-Functions in a Block</i>	246
<i>Automatic Corner Rounding</i>	247
PARAMETERS OF SPINDLE CONTROL	247
<i>Maximum Spindle Speed</i>	247
<i>Spindle Orientation Code</i>	247
PARAMETERS OF TOOL OFFSET	248
<i>Offset Address</i>	248

Lathe Offsets	248
Clearing Offsets	248
Radius vs. Diameter	248
Wear Offset	249
PARAMETERS OF CYCLES	250
Fanuc 15 Mode - Compatibility	250
Spindle Rotation Direction	250
Clearances in G73 / G83 Cycles	251
PARAMETERS OF THREADING CYCLE	251
PARAMETERS OF MULTIPLE REPETITIVE CYCLES	252
Available Cycles	252
Parameters for G71 / G72 Cycles	252
Parameters for G73 Cycle	253
Parameters for G74 / G75 Cycles	253
Parameters for G76 Cycle	253
BATTERY BACKUP	254
PARAMETERS AND G10	254
NO DECIMAL POINT ENTRY	255
Historical Background	255
Equivalent Numbers	256
Input Without a Decimal Point	256
SUMMARY	256
 Chapter 24	
PROGRAM OPTIMIZATION	257
PROGRAM REVIEW	257
PROGRAM RELATED CHANGES	257
CHANGES AT THE CONTROL	258
Spindle Speeds and Feedrates	258
Clearances	258
Optimized Program	259
Dwells	259
Combined Tool Motions	259
SETUP RELATED CHANGES	260
SUMMARY	260
 Chapter 25	
REFERENCES	261
G-CODES AND M-FUNCTIONS	261
Milling - G-codes	261
Milling - M-functions	262
G-CODES AND M-FUNCTIONS	262
Turning - G-codes	262
Turning - M-functions	263
DECIMAL EQUIVALENTS	263
PARAMETERS CLASSIFICATIONS	267
COMMON ABBREVIATIONS	269
TAP DRILL SIZES	271
Metric Threads - Coarse	271

<i>Metric Threads - Fine</i>	271
<i>Imperial Threads - UNC/UNF</i>	272
<i>Straight Pipe Taps - NPS</i>	273
<i>Taper Pipe Taps - NPT</i>	273
60-DEGREE THREAD FORMS	274
USEFUL FORMULAS	274
<i>Speeds and Feeds</i>	274
<i>Machining Holes - Spot Drilling</i>	275
<i>Tapers</i>	276
<i>Control Memory Capacity</i>	277
<i>Trigonometric Chart</i>	278
Index	279