

Outline for Getting More From Your CNC Machines Seminar

Advanced techniques with basic CNC features 11

- Benefits of knowing advanced techniques 12
- Machines covered 12
- Controls covered 12

Understanding parameters 13

- Eight bit binary type 13
 - Changing eight bit binary parameters 14
- Whole number type 14
- How parameters can affect machine behavior 14
 - A specific example 15
- Importance of backing up parameters 15
- Changing parameters 16
- A few more examples 16
 - How do you protect important programs from accidental deletion? 16
 - Finding the parameter 16
 - To protect programs in the 9000 series 16
 - Do you want to hide these programs? 16
 - Loading multiple programs from one file 17
 - What about saving multiple programs? 17
 - Limiting offset changes on turning centers 18
 - Finding the parameter 18
 - Changing initialized states 18
 - Two Parameters related to measurement systems 19

Documentation in a program 19

- Program headers 20
- Tool information 21
- At every program stop 22
- Simple setup instructions 23
- Describing changes made after a dispute 24
- Documenting something out of the ordinary 24
- Getting documentation into programs for controls that do not display messages 24

Using block delete 25

- Using block delete mid-command 26
- Conflicting words in a command 26
- Using block delete with unexpected rough stock 27
- Another optional stop 30
- Special note about multiple applications 30

Sequence number techniques 30

- Eliminating sequence numbers with limited memory capacity 31
- Using for restart blocks 31
- Sequence numbers as statement labels 32
 - Looping with block delete 32
 - Restarting after breaking a tap 33

Advanced discussion of G codes 35

- Testing for option G codes 35
 - Maximum number of G codes per command 35
- Ensuring initialized states with safety blocks 36
 - Monitoring the current state of G codes 37
 - Understanding G code groups 37
- G00 & G01 rapid and straight line motion for positioning (not cutting) 37
 - What is your maximum feedrate? 37
 - Which is initialized, G00 or G01? 38
 - Minimizing corner rounding 38
- G02 & G03 - circular motion commands 39
 - Which way is clockwise? 39
 - Limitation of the R word 40
 - Directional vectors 40
 - Arc limitations 41
 - Full circle in one command 42
- G02-G03 Helical interpolation (machining centers) 43
 - Helical interpolation for thread milling 43
 - Basic terminology 43
 - A test for helical interpolation 44
 - Thread milling cutters 45
 - Your approach to thread milling 46
 - Programming considerations 46
 - An example 47
 - Spiral interpolation for taper thread milling (machining centers) 49
 - Taper thread milling without spiral interpolation (or custom macro) 50
 - Cylindrical interpolation (machining centers) 51
 - Nurbs interpolation (machining centers) 54
 - Polar coordinate interpolation (turning centers) 55
 - Programming a rotary axis 57

Understanding polar coordinate interpolation 60	Using G32 for tapping 87
U axis on machining centers 63	Radius Compensation G40, G41, & G42 87
G04 – Dwell Techniques 65	The two ways to use offsets with cutter radius compensation 88
Can you specify the dwell period in number of spindle revolutions? 65	How cutter radius compensation works 89
Relieving tool pressure 65	Alarms and possible causes 91
Are you dwelling to overcome machine problems? 66	Diagnosing the over-cutting alarm 91
Poorly interfaced M codes 66	Insufficient clearance for the cutter at the starting position 91
G09 and G61 - exact stop check commands 67	Offset value is too large 92
G10 – Data Setting Command 67	Attempting to machine multiple contours under the influence of cutter radius compensation 93
How G10 works 68	Forgetting to cancel cutter radius compensation 93
Running out of fixture offsets? 68	Maximum cutter size exceeded alarm 94
What about reading offset values from within your programs? 69	Do you really need control based tool nose radius compensation (on turning centers)? 94
Setting parameters from within CNC programs 69	Machining on both sides of button tool 95
Running multiple workpieces from a single bar 70	Tool Length Compensation G43 95
G15 & G16 - polar coordinates for machining centers 71	Using secondary offsets with tool length compensation 95
Plane Selection G17, G18, & G19 72	G50 & G51 - Scaling 97
Plane selection with right angle heads 72	Spindle Limiter G50 (turning centers) 97
Circular commands 73	Chuck changes 98
Cutter radius compensation 74	Out-of-round workpieces 98
Using canned cycles with right angle heads 75	G50.1 & G51.1 - Mirror image commands 99
Other times you must specify plane selection 76	General explanation of mirror image 99
Milling with a ball end mill 76	Applications for mirror image 99
Plane selection with coordinate manipulation commands 77	The two ways to activate mirror image 99
Plane selection on turning centers 78	Manually turning on mirror image 99
G20 & G21 Inch / Metric Selection 78	Turning mirror image on in the program 99
How to select the inch or metric mode 79	Mirror image on a turning center? 100
Other considerations when switching to metric mode 80	G52 - temporary shift of program zero on machining centers 101
The advantage of the metric mode 80	G53 - rapid movement relative to the machine's zero return position 102
Selecting the mean value for a dimension 81	Another way to send the machine to the zero return position 103
Offset considerations 82	Use with manual pallet changers 103
How the control generates axis departure (inch versus metric mode) 82	Common turret index position on turning centers 103
G22 and G23 Stored stroke limit 84	G54-G59 - fixture offsets 103
How stored stroke limit works 85	The two ways to use fixture offsets 104
Setting up the safety zone 85	Using the machine's zero return position as the point of reference 104
G31 – Have You Ever Heard Of Touch Sensor? 86	Dealing with alignment problems after a crash 104
How it works 86	Using the common offset to handle machine differences 104
An application 86	
G32 - thread cutting command (turning centers) 87	

48 fixture offset option 105
 G60 - single direction positioning 105
 G64 - normal cutting mode 106
 G68 & G69 - coordinate rotation for machining centers 106
 G68 & G69 - Three dimensional coordinate conversion 107
 G70 - Turning center finishing cycle 108
 Using G70 to repeat commands 108
 G76 - threading cycle 110
 Maximum feedrate when threading 110
 What is thread chamfering? 110
 An unexpected problem caused by thread chamfering 111
 Specifying minimum depth of cut, final depth of cut, and number of spring passes 111
 Machining tapered threads 111
 Multiple start threads 112
 Using the E word when threading 114
 Canned Cycles G73-G89 114
 Parameters related to hole machining canned cycles 114
 Return amount in peck drilling cycle G73 114
 Clearance amount in deep hole peck drilling cycle G83 114
 L0 with canned cycles 115
 With subprogramming 115
 Using L0 to avoid obstructions 115
 Canned cycles in the incremental mode 116
 G90 & G91 - absolute and incremental mode 118
 Using both in one command 118
 Other applications for the incremental mode 119
 Moving relative to zero return 119

Advanced discussion of M codes 120

Reference your machine tool builder's programming manual 120
 Examples of M codes whose behavior you may not agree with 121
 Do you have some M codes that are not fully interfaced? 122
 M41 & M42 - spindle range changing on turning centers 122

Advanced discussion of T codes on turning centers 123

Right or left hand tools? 123
 Canceling wear offsets with T0 124
 Secondary applications for wear offsets 125

What happens when a wear offset gets changed? 125
 Applications for secondary wear offsets 125
 Flip jobs 126
 Holding size for critical with secondary wear offsets 127
 Using a secondary offset to allow for unwanted taper 128
 Using a secondary tool offset to machine long shafts 130
 Using a secondary offset to control groove width 132

Setup Reduction 135

Basic premises 135

CNC machine utilization versus application 135
 Machine utilization versus personnel utilization 136
 Criteria for wise decisions 137
 Company type 137
 Other factors contributing to a company's identity 138
 Company philosophies 142
 Watch out for conflicts in company type, company profile, and company philosophy! 143
 Which programming method should you use? 143
 Definitions for CNC machine usage 144
 The importance of value added principles in the CNC environment 147
 What's wrong with this picture? 148

Setup reduction principles 148

The importance of reducing setup time 148
 Should you reduce setup time or cycle time - or both? 149
 Justification for setup time reduction 150
 The relationship between production quantities, process, and setup time 151
 What are you trying to improve? 151
 Setup time defined 152
 Do you agree with our definition? 152
 The two types of setup tasks 153
 The three ways to reduce setup time 153
 Eliminate tasks 153
 Move tasks off line 154
 Facilitate tasks 154
 Convenience features versus setup time 155
 Available resources 156
 Ingenuity of your people 156

Purchased items 156	Eliminate cutting tool tasks 177
Thorough understanding of machine & control features and functions 156	Move cutting tool tasks off line 178
Parametric programming 156	Facilitate cutting tool tasks 182
Our suggested techniques 156	How tool life management systems affect setup time 185
Four steps to setup reduction 156	Cutting tools for turning centers 186
Step one: Evaluate your current methods 156	The difference between cutting tool setup and cutting tool maintenance 186
Step two: Brainstorm for improvements 157	Eliminate cutting tool tasks 186
Step three: Prioritize improvements and assign responsibilities 158	Move cutting tool tasks off line 187
Step four: Gauge success 158	Facilitate cutting tool tasks 188
Are you really ready to begin a setup reduction program? 159	Program zero assignment for machining centers 189
A suggestion for getting started 159	What are program zero assignment values? 189
Getting help 159	Eliminate program zero assignment tasks 190
Setup reduction techniques 160	How do you determine the program zero assignment values the first time a setup is made? 192
Preparation and organization for setup 161	How do you keep track of a job's program zero assignment values? 193
A well-designed work area 162	More on sub-plates 193
Other factors that contribute to a setup person's ability to be organized 162	Move program zero assignment tasks off line 195
Job order planning 163	Facilitate program zero assignment tasks 196
Standard tool stations 164	What if you don't have a spindle probe? 196
Taking advantage of non-production time 164	Program zero assignment for turning centers 198
Tasks related to preparing to make a specific setup 164	What are program zero assignment values? 198
Workholding setup for machining centers 165	Initially coming up with program zero assignment values 199
Eliminate workholding setup tasks 166	Eliminate program zero assignment tasks 199
Eliminate the need for squaring the workholding device with the table 166	Move program zero assignment tasks off line 200
Do you have any parameters to change? 168	Facilitate program zero assignment tasks 200
Move workholding setup tasks off line 169	Don't force setup people to reset all program zero assignments in every job 200
Facilitate workholding setup tasks 169	Use tool touch-off probes 201
Workholding setup for turning centers 169	Program development 202
A note about the workholding device 170	Remember! Programming inconsistencies can affect setup time 203
Eliminate workholding setup tasks 170	How many programs are we talking about? 203
If you can't eliminate all tasks related to setup, try to eliminate individual tasks. 170	Eliminate programming tasks 203
Do you have any parameters to change? 171	When is a program considered proven? 203
Move workholding setup tasks off line 171	Move programming tasks off line 204
Facilitate workholding setup tasks 172	Facilitate programming tasks 205
How hard is it to replace top tooling? 172	
What about quick-change chucks? 173	
Facilitating the task of jaw machining 173	
Why not program the jaw boring process? 175	
Cutting tools for machining centers 176	
The difference between cutting tool setup and cutting tool maintenance 177	

Do you run parts in a family? 205	We cannot stress enough the importance of knowing your current costs 233
The need for program transfer 206	The relationship between production quantities, process, and cycle time 233
Eliminate tasks related to program transfer 207	What are you trying to improve? 234
Move program transfer off line 207	Your available resources 234
Facilitate program transfer 207	Cycle time defined 235
Use a DNC system 207	Two task types 236
Program verification 208	On-line tasks 236
Tasks related to program verification 208	Off-line tasks 236
How important is it that your first workpiece be a good one? 209	Warning! 236
Eliminate program verification tasks 210	Productive versus non-productive tasks 236
When is a program considered proven? 210	Three ways to reduce cycle time 236
How are you handling other setup-related tasks? 211	Eliminate tasks (both on-and off-line tasks) 236
Move program verification tasks off line 212	Move on-line tasks off line 237
Facilitate program verification tasks 212	Facilitate tasks (both on- and off-line tasks) 237
Facilitate the task of trial machining 213	Four steps to cycle time reduction 237
Here are some examples of trial machining. 214	Step one: Evaluate your current methods 238
Eliminating tool pressure problems when finish turning or boring 216	Step two: Brainstorm for improvements 238
Facilitating the sizing for lengthy threading operations 218	Step three: Prioritize improvements and assign responsibilities 239
Facilitating finish boring on machining centers 219	Step four: Gauge success 240
Other potential applications for trial machining 220	Think you're ready to start? 240
Facilitate process (machining order) changes 220	A suggestion for getting started 241
What about first workpiece inspection? 223	Which task do you concentrate on first? 241
Eliminate first workpiece inspection tasks 223	The one-second rule 242
Move first workpiece inspection tasks off line 223	Other time related formulae 242
Facilitate first workpiece inspection tasks 223	How fast can your machines rapid? 242
Program optimizing 224	Getting help 243
The one-second rule 224	Cycle time reduction techniques 244
Eliminate program-optimizing tasks 224	Preparation and organization for running production 244
Move program-optimizing tasks off line 224	A well-designed work area 246
Facilitate program-optimizing tasks 225	Other factors that contribute to an operator's ability to be organized 246
Other tasks related to making setups 226	Tasks related to preparing to run a specific job 248
	Should the operator be doing these tasks? 248
	Can you eliminate any of these tasks? 248
	What tasks can be done off line? 248
	Get everything the operator needs at one time 248
	Facilitate organization tasks 248
	Workpiece loading for machining centers 249
	Eliminate workpiece loading tasks 249
	Eliminate workpiece preparation 249
Cycle Time Reduction 231	
Cycle time reduction principles 231	
The importance of reducing cycle time 231	
Should you reduce cycle time or setup time - or both? 232	
Justification for cycle time reduction 233	

- What else can you eliminate? 250
- Move workpiece loading tasks off line 250
 - Use pallet changers 250
 - What about manual pallet changers? 250
 - Keep all workpiece preparation off line 250
- Facilitate workpiece loading tasks 250
- Use machines with automatic doors 251
 - Quality of work area facilitates preparation 251
 - What kind of workholding device are you using? 251
 - Watch out for other tasks that creep into on-line workpiece loading 251
- Workpiece loading for turning centers 252
 - Eliminate workpiece loading tasks 252
 - Eliminate workpiece preparation 252
 - Eliminate the need to inspect loaded workpieces 253
 - What else can you eliminate? 253
 - Move workpiece loading tasks off line 253
 - Keep all workpiece preparation off line 253
 - Facilitate workpiece loading tasks 253
 - Use machines with automatic doors 253
 - Quality of work area facilitates preparation 254
 - What kind of workholding device are you using? 254
 - Watch out for other tasks that creep in to on-line workpiece loading 254
- Program execution (common to machining centers and turning centers) 254
 - The importance of reducing program execution time 255
 - How good is your machining process 256
 - Machine functions that affect cycle time 256
 - Spindle acceleration and deceleration 256
 - Axis acceleration and deceleration 256
 - M code activation 256
 - In position check 257
 - Tool changing functions 257
 - Activation of other machine functions 257
 - Higher level programming features 258
 - Parameters related to program execution time 259
 - G73 chip break peck drilling cycle 259
 - G83 deep hole peck drilling cycle 259
 - G71 rough turning cycle 259
 - Where do you find parameter documentation? 260
 - Program structure 260
 - Include M codes with motion commands 260
 - Do as much as you can in each command 260
 - Minimize rapid motions 261
 - Keep tool change position close to the workpiece 261
 - Cautiously watch your programs execute! 261
 - Reduce rapid approach distance 262
 - How much did we save? 265
 - Safety implications 265
 - By the way, what is an appropriate approach distance? 265
 - Which is faster, G00 or G01? 266
 - Having trouble believing it? 268
 - Which method does G81 use? 268
- Program execution (machining centers) 269
 - Eliminate program execution tasks 269
 - Machining process is followed 269
 - The CNC control sequentially executes the program 270
 - Tool changing 270
 - We wish to run 5 workpieces per cycle instead of one 271
 - Minimize spindle range changes during tool changing 271
 - Rapid approach in all axes 272
 - Eliminate unnecessary movements 272
 - Do you have any manual intervention? 272
 - Move machining operations off line 272
 - Be sure safety commands are executed off line 273
 - Double up on machine functions 273
 - Do you have any manual intervention? 274
 - Facilitate program execution tasks 275
 - Treat manual intervention like workpiece loading 275
- Program execution (turning centers) 275
 - Eliminate program execution tasks 276
 - No need for polishing after cycle 276
 - The CNC control sequentially executes the program 276
 - Never use the dwell command to program around machine problems! 276
 - Turret indexing 277
 - Minimize spindle range changes during tool changing 277
 - Incorporating a flexible turret index position for turning centers 278
 - Minimize spindle direction reversals 280
 - Minimize G01 positioning movements 281
 - Eliminate unnecessary movements 281
 - Do you have any manual intervention? 281
 - Move program execution tasks off line 282
 - Move machining operations off line 282
 - Be sure safety commands are executed off line 282
 - Move spindle dead time off line 283
 - Double up on machine functions 286
 - Do you have any manual intervention? 287

Facilitate program execution tasks 287	To eliminate calculations prior to offset entry on turning centers 300
Can your bar feeders keep up with the machine's spindle? 287	What is the target dimension? 300
Workpiece unload 291	Minimizing repeated calculations 300
Eliminate workpiece unloading tasks 291	What about more complicated offset entries? 302
Move workpiece unloading off line 291	Use offsets for sizing adjustments whenever possible 303
Facilitate workpiece unloading tasks 291	What about mechanical size holding adjustments? 304
Off line tasks done during every cycle 292	Facilitating workpiece sizing tasks 305
Where is the constraint? 292	What do your operators use to measure? 306
What is the constraint of the CNC operation? 293	How good is your production run documentation? 306
Operations on the completed workpiece 293	Can design engineers help? 306
Why are you doing this? 293	Can programmers help? 307
What about manufacturing cells? 294	Automatic workpiece sizing systems 307
Eliminate, move off line, facilitate 294	Dull cutting tool replacement (tool maintenance) 308
Workpiece inspections done during the CNC cycle 295	When is tool replacement required? 308
Sizing (offset) adjustments due to tool wear 295	Eliminate these tasks? 309
The difference between initial adjustment, sizing adjustments, and tool replacement 295	Move tool maintenance tasks off line 310
More on sizing 296	Can you be removing and replacing tools while the machine is in cycle? 310
The entire task of sizing should be off line! 297	Can you justify duplicating tools? 310
What offsets are you using for sizing? 297	Do you have a tool life management system? 310
Turning center users! Are you using offsets appropriately? 297	Move tool preparation off line 311
Eliminate workpiece sizing? 298	Facilitating dull tool replacement 311
What are you shooting for? 298	Does your turning center have a tool touch-off probe? 312
What is the deviation? 299	How accurate are your inserts? 312
What is the polarity of the deviation? 299	
Make the offset entry 299	
Eliminate all calculations! 299	