

# Course outline

## **Know Your Machine From A Programmer's Viewpoint 11**

If you've had experience with conventional (non-CNC) machine tools... 11

## **Machine Configurations 13**

Vertical machining centers 13

C-frame style 13

Directions of motion (axes) for a C-frame style vertical machining center 14

Axis polarity 14

Knee style vertical CNC milling machines 15

Bridge-style vertical machining center (also called gantry-style) 16

Horizontal machining centers 17

Directions of motion (axes) for a horizontal machining center 18

Axis polarity 18

Programmable functions of machining centers 20

Spindle 20

Spindle speed 20

Spindle activation and direction 20

Spindle range 20

Feedrate 21

Coolant 22

Automatic tool changer 22

Measurement system mode (inch or metric) 23

What else might be programmable? 24

Key points for Lesson One: 24

## **General Flow Of The CNC Process 25**

Companies that use CNC machining centers 25

What will you be doing? 26

Flow of the CNC process 26

Study the workpiece drawing 26

Decision is made as to which CNC machine to use 26

The machining process is developed 27

Tooling is ordered and checked 27

The program is developed 27

Setup and production run documentation is made 27

Program is loaded into the CNC control's memory 27

The setup is made 27

The program is cautiously verified 28

Production is run 28

Corrected version of the program is stored for future use 28

Key points for Lesson Two: 28

## **Visualizing The Execution Of A CNC Program 29**

Program make-up 30

Method of program execution 30

An example of program execution 30

Manual milling machine procedure: 31

CNC program: 31

Sequence numbers 32

A note about decimal point programming 32

A decimal point tip 33

Other mistakes of omission 33

Modal words 33

Initialized words 33

Letter O or number zero? 33

Word order in a command 34

Key points for Lesson Three: 34

## **Program Zero And The Rectangular Coordinate System 35**

Graph analogy 35

What about the Z axis? 36

Understanding polarity 37

Wisely choosing the program zero point location 40

In X and Y 40

Reminder about axis movement 41

In Z 41

Absolute versus incremental positioning modes 43

A decimal point reminder 44

Key points for Lesson Four: 44

## **Determining Program Zero Assignment Values 45**

Program zero assignment values 45

What is zero return position? 46

Determining program zero assignment values 47

Manually measuring the program zero point location on the machine 47

In X and Y for a square or rectangular workpiece 47

Key points for Lesson Five: 49

## **Assigning Program Zero 51**

Understanding fixture offsets 51

Key points for Lesson Six: 52

**Introduction To Programming Words 53**

Words allowing a decimal point 53

O 54

N 54

G 54

X 54

Y 54

Z 54

A 54

B 55

C 55

Note about rotary axis designators and indexer activators 55

R 55

I, J, K 55

Q 56

P 56

L 56

F 56

S 56

T 57

M 57

D 57

H 57

EOB (end of block character) 57

/ (slash code) 57

G and M codes 58

G codes 58

G code limitation: 58

Option G codes 58

What does initialized mean? 58

What does modal mean? 58

The most popular G codes 58

Common M codes used on a CNC machining center 59

Key points for Lesson Seven: 60

**You Must Prepare To Write Programs 61**

Preparation and time 61

Preparation and safety 62

**Preparation Steps For Programming 63**

Prepare the machining process 63

Develop the needed cutting conditions 65

An example 66

Cutting conditions can be subjective 66

Do the required math and mark-up the print 67

Marking up the print 67

Doing the math 68

What about milling operations? 70

Check the required tooling 71

Plan the work holding set-up 72

Other documentation needed for the job 73

Production run documentation 74

Program listing 74

Is it all worth it? 74

Key points for Lesson Eight: 74

**Understand The Motion Types 75**

What is interpolation? 75

**Programming The Three Most Basic Motion Types 79**

Motion commonalties 79

Understanding the programmed point of each cutting tool 79

Center drill 80

Spot drill (not shown above) 80

Drill 80

Reamer 80

Tap 80

Boring bar 80

What about milling cutters? 81

G00 – Rapid motion (also called positioning) 82

How many axes can be included in a rapid motion command? 83

About the dog-leg motion... 83

When do you use rapid motion? 84

What is a safe approach distance? 84

G01 – Linear interpolation (straight-line motion) 85

Using G01 for a fast-feed approach 86

A milling example 87

Drill holes with G01? 87

G02 and G03 – Circular interpolation (circular motion) 88

Which positions to program 88

Specifying arc size with the R word 89

The R word is not modal 90

More details about clockwise versus counter clockwise circular motion 90

Planning your own tool paths 91

Key points for Lesson Nine: 91

**Know The Compensation Types 93**

**Introduction To Compensation 95**

What is compensation and why is it needed? 95

More on tolerances 96

- The initial setting for compensation 96
- When is trial machining required? 96
- What happens as tools begin to wear? 97
- What do you shoot for? 97
- Why do programmers have to know this? 97
- Understanding offsets 97
  - Offset organization 98
    - Offsets related to cutting tools 98
    - Offsets related to program zero assignment 99
  - How offsets are instated 99
- Key points for Lesson Ten: 100

### ***Tool Length Compensation 101***

- The reasons why tool length compensation is needed 101
  - No two tools will have exactly the same length 101
  - A given tool's length will vary from one time it is assembled to the next 102
  - Tool data is entered separately from the program 102
  - Sizing and trial machining must often be done 102
    - What about interference and reach? 102
- Programming tool length compensation 102
  - Choosing the offset number to be used with each tool 102
  - An example program 103
- The setup person's responsibilities with tool length compensation 104
  - Recommended method: Using the tool's length as the tool length compensation (offset) value 104
    - Determining tool length compensation values 105
    - Entering tool length compensation offsets 107
  - Using the distance from the tool tip to program zero as the tool length compensation (offset) value 108
    - Measuring tool length compensation values with this second method 108
- Reasons for using our recommended method 109
- Typical mistakes with tool length compensation 109
  - Forgetting to instate tool length compensation 109
  - Forgetting to enter the tool length compensation value 110
  - Mismatching offsets 110
- Trial machining with tool length compensation 110

- When trial machining is not required 110
- Sizing with tool length compensation 111
  - A tip for remembering which way to adjust the offset 111
  - What if I use the second method shown for tool length compensation? 111
  - Do I have to make all these calculations when adjusting offsets? 111
  - Why can't I just change the Z coordinate/s in the program to make sizing adjustments? 111
- Key points for Lesson Eleven: 112

### ***Cutter Radius Compensation 113***

- Reasons why cutter radius compensation is required 113
  - Calculations are simplified for manual programmers 113
  - Range of cutter sizes 115
    - Do you use re-sharpened (re-ground) cutters? 116
  - Trial machining and sizing 117
  - Rough and finish milling with the same set of coordinates 117
- How cutter radius compensation works 118
- Steps to programming cutter radius compensation 119
  - Step one: Instate cutter radius compensation 119
    - The XY motion to the prior position 119
    - The Z motion/s to the Z axis work surface 121
  - The command instating cutter radius compensation that positions the cutting tool to the first surface to mill 121
    - G41 or G42? 121
    - The offset used with cutter radius compensation 122
    - The motion to the first work surface 123
  - Step two: Program the tool path to be machined 124
  - Step three: Cancel cutter radius compensation 126
    - What if I have more than one contour to mill? 127
- Example 128

### ***Fixture Offsets 129***

- Assigning multiple program zero points 129
- Key points for Lesson Thirteen: 132

## **You Must Provide Structure To Your CNC Programs 133**

### ***Introduction To Program Structure 135***

Objectives of your chosen program structure	135
Reasons for structuring programs with a strict and consistent format	135
Familiarization	135
Consistency	136
Re-running tools in the program	136
Efficiency limitations	137
Machine variations that affect program structure	137
M code differences	137
Automatic tool changer commands	138
T word brings a tool to the ready station, M06 commands the tool change	138
Tool change at beginning or end?	140
Understanding the G28 command	140
A possible problem with initialized modes	141
How to use our given formats	142

### **Four Types Of Program Format 143**

Format for vertical machining centers	143
Program Start-Up Format:	144
Tool Ending Format:	144
Tool Start-Up Format:	144
Program Ending Format:	144
N001	144
N002	145
N003	145
N005	145
N010	145
N015	145
N020	145
N025	145
N030	145
N075	145
N080	145
N085	146
N135 - N160	146
N310 - N320	146
N325	146
A note about documentation	146
Example program for vertical machining centers	147
Key points for Lesson Fifteen:	148

### **Special Features That Help With Programming 149**

### **Hole-Machining Canned Cycles 151**

Canned cycle commonalities	152
Description of each canned cycle	152
G80 – Cancel the canned cycle mode	152
G81 – Standard drilling cycle	152
G73 – Chip-breaking peck drilling cycle	152
G83 – Deep-hole drilling cycle (full retract between pecks)	152
G84 – Right-hand tapping cycle	153
G74 – Left-hand tapping cycle	155
G82 – Counter-boring cycle	155
G89 – Counter-boring cycle for a boring bar	155
G86 – Standard boring cycle (leaves drag line witness mark)	156
G76 – Fine boring cycle (leaves no witness mark)	156
G85 – Reaming cycle (most programmers use G81 for reaming)	158
G87 and G88 – Manual cycles (not recommended)	158
Words used in canned cycles	158
A simple example	158
Understanding G98 and G99	160
Canned cycles and the Z axis	162
Extended example showing canned cycle usage	164
N020, N105, N175	167
N030, N115, N185, N255, N300, N344, N395	167
N245, N290, N335	167
N255, N300, N345	167
N045, N050, N125, N130, N195, N200	167
N075, N145, N215, N260, N305, N355, N405	167
N255	167
N300	167
N185	168
Using canned cycles in the incremental positioning mode	168
Key points for Lesson Sixteen:	170

### **Working With Subprograms 173**

The difference between main- and sub-programs	173
Words used with subprograms	175

### **Know Your Machine From An Operator's Viewpoint 176**

The need for hands-on experience	176
Procedures you must know about	177

Machine power-up 177	Activate the cycle 198
Sending the machine to its zero return position 177	Monitor the cycle 199
Manually moving each axis 177	Clean and de-burr the workpiece 199
Manually starting the spindle 177	Perform specified measurements 200
Manually making tool changes 178	Make offset adjustments to maintain size for critical dimensions (sizing) 200
Manipulating the display screen 178	Replace worn tools 201
The position display screen 178	Clean the machine 201
The program display screen 178	Preventive maintenance 201
<b>Tasks Related To Setup And Running Production 179</b>	Key points for Lesson Twenty: 201
A CNC job from start to finish 181	<b>Buttons And Switches On The Operation Panels 203</b>
Setup documentation 182	The two most important operation panels 203
Tear down the previous setup and put everything away 183	The control panel buttons and switches 203
Gather the components needed to make the setup 183	Display screen control keys (soft keys) 204
Make the workholding setup 183	Position display pages 205
Assign the program zero point 184	Program display pages 205
Measure program zero assignment values 184	Offset display pages 205
Enter program zero assignment values into fixture offset number one 186	Program check display pages 206
Assemble the cutting tools needed for the job 186	Other display screen modes 206
Measure tool length- and cutter-radius-compensation values 187	The keyboard 207
The static nature of cutting tool measurements 188	Letter Keys 207
Enter tool length and cutter radius compensation values into offsets 188	The slash key (/) 207
Load cutting tools into the machine's automatic tool changer magazine 188	Number keys 207
Load the CNC program 189	Decimal point key 207
Verify the correctness of a new or modified program 189	The input key 207
Verify the correctness of the setup 190	Cursor control keys 207
Dry running our example program 191	Program Editing Keys 207
Cautiously run the first workpiece 192	Reset key 207
The most dangerous time 193	The machine panel 208
Making sure the first workpiece is a good one 194	Mode switch 208
If necessary, optimize the program for better efficiency 196	Cycle start button 208
If changes have been made to the program, save the corrected version of the program 196	Feed hold button 208
Production run documentation 197	Feedrate override switch 209
Remove the previous workpiece 198	Rapid traverse override switch 209
Load the next workpiece 198	Emergency stop button 209
	Conditional switches 210
	Dry run on/off switch 210
	Single block on/off switch 210
	Block delete on/off switch (also called optional block skip) 210
	Optional stop on/off switch 210
	Buttons and switches for manual functions 210
	Axis jogging controls 211
	Handwheel controls 211
	Spindle control 211
	Automatic tool changer control 211
	Indicator lights and meters 212
	Spindle rpm and horsepower meters 212

Axis drive-motor horsepower meter 212  
 Cycle indicator lights 212  
 Zero return position indicator lights 212  
 Optional stop indicator light 212  
 Other buttons and switches on the machine panel 212  
 Other operation panels on your machining center 212

## **Know The Three Basic Modes Of Operation 213**

### ***The Three Modes Of Operation 213***

The manual mode 213  
 The manual data input mode 214  
 The manual data input (MDI) mode switch position 214  
 Commanding an MDI zero return 214  
 The complete procedure to give an MDI command 215  
 Commanding an MDI tool change 215  
 Commanding spindle activation with MDI 215  
 Other times when MDI is used 216  
 Can you make motion commands with MDI? 216  
 The edit mode switch position 216  
 To make a program in memory the active program (to call up a program) 217  
 To enter a new program 217  
 The program operation mode 218  
 To run the active program from the beginning 218  
 Key points for Lesson Twenty-Two: 218

## **Understand The Importance Of Procedures 219**

### ***The Key Operation Procedures 220***

Key points for Lesson Twenty-Three: 224

## **You Must Know How To Safely Verify Programs 225**

Safety priorities 225  
 Operator safety 225  
 Machine tool safety 226  
 Workpiece safety 226

### ***Program Verification 227***

Two more procedures 227  
 Canceling the CNC cycle 227  
 To re-run a tool 228

# Turning Center Discussions Start Here

## **CNC Turning Centers 230**

### **Know Your Machine From A Programmer's Viewpoint 231**

If you've had experience with conventional (non-CNC) machine tools... 231

### ***Machine Configurations 233***

Types of CNC turning centers 233  
 Universal style slant bed turning center 233  
 Directions of motion (axes) for a universal style slant bed turning center 234  
 Live tooling for a universal style slant bed turning center 235  
 Programmable functions of turning centers 237  
 Spindle 237  
 Spindle speed 237  
 Spindle activation and direction 237  
 Spindle range 238  
 Feedrate 239  
 Turret indexing (tool changing) 240  
 Turret station and offset selection 241  
 Coolant 241  
 Other possible programmable functions 241  
 Tailstock 241  
 What else might be programmable? 242  
 Key points for Lesson One: 242

### ***Understanding Turning Center Speeds and Feeds 243***

The two ways to select spindle speed 244  
 When to use constant surface speed mode 245  
 When to use rpm mode 246  
 How to specify a maximum speed for the constant surface speed mode 248  
 The two ways to specify feedrate 248  
 When to use the feed per revolution mode 249  
 When to use the feed per minute feedrate mode 249  
 An example of speed and feed usage 250

### ***Program Zero And The Rectangular Coordinate System 251***

Graph analogy 251

More about polarity	252	Offset pages on the display screen	296
Wisely choosing the program zero point location	253	How offsets are instated	298
In X	253	<b>Wear offsets 299</b>	
In Z	253	Which dimension do you choose for sizing?	300
<b>Determining Program Zero Assignment Values 258</b>		How wear offsets are programmed	300
Program zero must be assigned independently for <i>each</i> cutting tool	258	What if my machine doesn't have geometry offsets?	300
Understanding program zero assignment values	260	What about wear offset cancellation?	301
What is the zero return position?	260	How wear offsets are entered	302
Program zero assignment values	260	Which is better, INPUT or +INPUT?	304
How do you determine program zero assignment values?	262	What if my machine doesn't have a +INPUT soft key?	304
<b>Assigning Program Zero 263</b>		Sizing in a tool after it has just been placed in the turret	305
Understanding geometry offsets	263	Sizing in a new tool with trial machining	306
How geometry offsets are instated	264	What causes the initial deviation?	306
The four most common ways to assign program zero	265	Dealing with deviations caused by tool wear	307
Determining and entering the work shift value	267	After a dull tool is replaced	308
Using geometry offsets with work shift to assign program zero	269	Consistently replacing inserts	309
Understanding the measure function	270	Consistently indexing inserts	310
<b>You Must Prepare To Write Programs 272</b>		<b>Tool Nose Radius Compensation 311</b>	
<b>Preparation Steps For Programming 273</b>		Keeping the cutting edge flush with the work surface at all times	313
Doing the math	273	When to use tool nose radius compensation	314
Check the required tooling	276	Steps to programming tool nose radius compensation	314
<b>Understand The Motion Types 281</b>		Instating tool nose radius compensation	314
What is interpolation?	281	Programming motion commands to machine the workpiece	315
<b>Programming The Three Most Basic Motion Types 284</b>		Canceling tool nose radius compensation	316
Understanding the programmed point of each cutting tool	284	An example program	316
G00 Rapid motion (also called positioning)	287	Tool nose radius compensation from a setup person's point of view	317
G01 linear interpolation (straight line motion)	288	<b>You Must Provide Structure To Your CNC Programs 321</b>	
G02 and G03 Circular motion commands	290	<b>Introduction To Program Structure 322</b>	
Specifying a circular motion with the radius word	291	Objectives of your program structure	322
<b>Know The Compensation Types 295</b>		Machine variations that affect program structure	322
<b>Introduction To Compensation 296</b>		M code differences	323
Understanding offsets	296	G code numbering differences	324
Offset organization	296	Turret variations	324
		Choosing the appropriate spindle range	325
		Which direction do you run the spindle?	325

How do you check what each tool has done?  
325

**Four Types Of Program Format 327**

Format for assigning program zero with geometry offsets 328

Program startup format (using geometry offsets) 328

Tool ending format (using geometry offsets) 329

Tool startup format (using geometry offsets) 329

Program ending format (using geometry offsets) 329

Example programs showing format for turning centers 330

Example when assigning program zero with geometry offsets 330

Where are the restart commands? 331

**Special Features That Help With Programming 335**

Control series differences 336

**G71 And G70 – Rough Turning And Boring Followed By Finishing 337**

G71 - Rough turning and boring 337

The two phases of G71 338

Understanding G71 command words 340

P word 340

Q word 340

U word 340

W word 340

D word 340

F word 341

What about finishing? 341

Example showing G71 for rough turning and G70 for finish turning 341

**G76 – Threading Command 345**

X word 345

Z word 346

What is thread chamfering? 346

K word 346

D word 346

A word 347

F word and E word 347

I word 347

Q word 347

Example program for threading 348

**INDEX 351**