

Table of Contents

PREFACE	11	Who creates a formula?	29
Prerequisites	11	More about numbers	30
Basic machining practice experience	12	Integers.....	30
Math.....	12	Real numbers.....	30
Motivation.....	12	Decimal format.....	30
Controls covered	12	Fractional format.....	30
Limitations	12	Converting fractional format to decimal format	31
The need for hands-on practice	12	Decimal equivalents for common fractions ...	32
Instruction method	13	Decimal places.....	32
Scope	13	Rounding to a given number of decimal places	32
Key Concepts approach.....	13	Measurement systems	33
Lesson structure	14	Saying numeric values out loud in a machine shop	34
Practice makes perfect.....	14	Saying values in the Metric system	35
Key Concepts and lessons	14	Understanding polarity.....	35
Enjoy!	14	Summary of shop math	35
 		Interpreting an engineering drawing	37
GET READY TO LEARN ABOUT CNC		Orthographic projection.....	37
MACHINING CENTERS	15	Line appearance	38
Basic Machining Practices Required for CNC		Non-standard views.....	39
Machining Centers	17	Isometric views.....	39
Shop Safety	17	Angular views	40
Safety equipment	17	Detail views	40
Safety practices	18	Section views.....	41
Handling raw material	19	Multiple page engineering drawings.....	42
Finished workpiece handling.....	19	Do you have a sample workpiece?	42
Tightening and loosening fasteners	19	Drawing scale	42
Getting around the shop	20	Dimensioning.....	43
Behavior	20	Measurement systems	44
Machining-center-specific safety practices....	20	Other descriptive notes.....	44
Shop math	21	Title block	44
A few suggestions for choosing your calculator.....	21	Revisions.....	44
Keep it simple	21	Introduction to tolerance interpretation	46
Watch out for trick functions	21	Who specifies tolerance?	47
Avoid solar powered calculators	22	Dimensional tolerances	48
Big buttons – big display	22	Tolerance terminology	49
Be sure it has clear entry button.....	22	Judging acceptability	50
Calculator basics.....	23	What are you shooting for?.....	50
What is an arithmetic expression?.....	23	Calculating the deviation (adjustment)	
Practice makes perfect	24	amount	50
The most basic arithmetic functions	24	Determining the deviation (sizing	
Addition.....	24	adjustment) polarity.....	50
Subtraction	25	More on dimensional tolerance specifications	50
Multiplication	25	High/low limits	51
Division	25	Uneven +/-.....	51
More on the priority of arithmetic operators	25	You may get some help	52
The use of parentheses in an expression	25	Implied tolerances.....	52
Does your calculator have parentheses? ...	26	Four conditions of a measured surface.....	52
Does your calculator have one or more		If the surfaces is not within low and high	
memories?.....	26	limits, can the workpiece be saved?.....	52
What is a formula?	28	Tool wear affects acceptability.....	53

Cutting tool type affects how quickly tool wear affects machined surfaces	53	Summary	96
Dealing with deviations.....	54	What Does A CNC Machining Center Do?.....	99
Choosing the surface to measure	54	Comparing a CNC machining center to other types of machines	99
More on determining deviation amount ...	55	Milling machines	100
Setup sizing versus production run sizing.....	55	Machining operations performed on machining centers	102
During setup.....	55	Introduction to cutting conditions	102
During production runs.....	56	Factors that contribute to cutting condition selection	102
Why targeting the mean value may not be best	56	Cutting tool materials	103
Geometric (surface relationship) tolerances	57	Roughing versus semi-finishing versus finishing.....	103
Measuring devices	60	How cutting conditions are recommended .	103
Checking calibration.....	60	Spindle speed: Converting from surface feet per minute (sfm) to revolutions per minute (rpm)	104
Measurement systems affect gauges	60	Spindle speed: Converting from meters per minute (mpm) to revolutions per minute (rpm)	104
Gauge resolution	61	Feedrate: Converting inches per tooth (ipt) or inches per revolution (ipr) to inches per minute (ipm)	104
Common gauge resolutions in each measurement system	61	Feedrate: Converting millimeters per tooth (mmpt) or millimeters per revolution (mmpr) to millimeters per minute (mmpm)	104
Environments for measuring	61	Getting recommendations for cutting conditions.....	105
Minimizing the impact temperature changes	63	Other cutting conditions related to machining operations	105
Measurement types.....	63	Recognizing dull tools	105
Gauge types	65	Hole-machining operations	106
Fixed gauges.....	66	Center drilling	107
Go/No-go fixed gauges	66	What about chamfering the hole?	108
Fixed gauges with dial displays	67	Cutting conditions for center drilling	108
Importance of the secondary dial.....	71	Dull tool recognition and sharpening.....	108
More on needle rotation direction and internal versus external measurements.....	71	Spot drilling.....	108
Variable gauges.....	74	Cutting conditions for spot drilling	109
Display types	76	Dull tool recognition and sharpening.....	110
Electronic digital display	76	Counter sinking	110
Dial display	77	Cutting conditions for counter sinking.....	111
Dial indicators	79	Dull tool replacement and sharpening.....	111
Vernier display	81	Drilling with twist drills	111
Display type demonstrations	82	Allowing for the drill point	112
Accuracy of the measuring device	82	Calculating the depth of through holes	112
Checking calibration.....	82	Peck drilling to break chips	112
A general suggestion for getting started with any variable gauge	84	Peck drilling for deep holes to clear chips ...	112
Interpreting (reading) the displayed value of a Vernier micrometer	85	Cutting conditions for twist drills.....	113
General use of a micrometer that has a clamp and ratchet	86	Applications for drilling with twist drills	113
Steps to reading the displayed value for a micrometer that measures in inches.....	86	Dull tool recognition and sharpening.....	113
Steps to reading the displayed value for a micrometer that measures in millimeters .	88	Other drilling tools	114
Interpreting (reading) the displayed value of a dial caliper.....	91	Spade drills.....	114
Steps to reading the displayed value for a micrometer that measures in inches	91	Carbide inserted drills.....	114
Steps to reading the displayed value for a micrometer that measures in millimeters .	92	Dull tool recognition and sharpening.....	115
Interpreting (reading) the displayed value of a Vernier caliper.....	93	Counter boring.....	115
Using a depth gauge	95		
Testing for calibration	96		

Cutting conditions for counter boring tools . 116	Setup and production run documentation is made	143
Dull tool sharpening	Program is loaded into the CNC control's memory	143
Reaming..... 116	The setup is made	144
Cutting conditions for reamers	The program is cautiously verified (still considered part of setup)	144
Applications for reaming	Production is run	144
Boring with boring bars..... 117	Corrected version of the program is stored for future use	144
Cutting conditions for boring bars		
Dull tool recognition and sharpening..... 118		
Tapping..... 118	Machine Configurations..... 147	
Tapping with tension-compression tap holders versus rigid tapping	Vertical machining centers	147
More about taps..... 119	C-frame style	147
The hole before tapping..... 120	Directions of motion (axes) for a C-frame style vertical machining center	148
Actions required for tapping	Axis polarity	148
Cutting conditions for taps..... 121	Knee style vertical CNC milling machines.....	150
Dull tool recognition and sharpening..... 122	Bridge-style vertical machining center (also called gantry-style)	151
Milling operations	One more point about the confusion related to axis polarity	151
Milling cutters that can plunge like hole machining tools	Horizontal machining centers	152
Climb versus conventional milling..... 124	Directions of motion (axes) for a horizontal machining center	153
Radial and axial depth of cut..... 125	Programmable functions of machining centers	154
Rough milling versus finish milling	Spindle.....	154
Cutting tool material	Spindle speed	155
Cutting conditions for milling operations..... 127	Spindle activation and direction.....	155
Types of milling cutters	Giving a spindle command	156
End mills	Coolant	156
Circle milling a counter bored hole	Automatic tool changer	156
Pocket milling	What else might be programmable?.....	157
Keyway and slot milling.....	Other important machine functions	157
Face milling.....	Motion rate	157
Exterior peripheral milling.....	Mode switch	158
Face mills.....	More on the reference position	158
Other milling cutters		
Ball and bull end mills	Buttons And Switches on the Operation Panels.... 161	
Slotting cutters	The Two Most Important Operation Panels	161
Thread milling cutters	The Control Panel Buttons and Switches	162
Motions needed for thread milling	Display Screen Mode Keys.....	163
Machining the entire thread in one pass	Position Display Pages	164
Summary of key points.....	Program Display Pages	164
136	Offset Display Pages	165
Conclusion to Key Concept Number One	Graph Display Pages	166
136	Other Display Screen Modes	166
	The Keyboard	166
KNOW YOUR MACHINE FROM AN OPERATOR'S VIEWPOINT..... 139	Letter Keys	167
The need for hands-on experience	The Slash Key (/)	167
139	Number Keys	167
	Decimal Point Key.....	167
General Flow of the CNC Process	CAN Key	167
141	EOB Key	167
Companies that use CNC machining centers	The INPUT Key	167
What will you be doing?.....		
142		
Flow of the CNC process		
143		
Study the workpiece engineering drawing.....		
143		
Decision is made as to which CNC machine to use		
143		
The machining process is developed.....		
143		
Tooling is ordered and checked		
143		
The program is developed.....		
143		

Cursor Control Keys	167	MDI Procedures	182
Program Editing Keys	167	Program Manipulation Procedures.....	182
RESET Key.....	167	Setup Procedures	182
The Machine Panel Buttons and Switches.....	168	Program Running Procedures	182
Power Buttons	168	Blank Procedure Form	183
MODE switch.....	168	Sample Operation Handbook: Vertical	
CYCLE START Button	169	Machining center with a 0iF FANUC CNC	184
FEED HOLD Button	169	Manual Procedures.....	184
FEEDRATE OVERRIDE Switch.....	169	To Power-Up the Machine	184
RAPID OVERRIDE Switch	169	To Do a Reference Return (send each axis to its	
EMERGENCY STOP Button	170	reference position).....	184
Conditional Switches.....	170	To Start the Spindle.....	184
DRY RUN On/Off Switch.....	170	To jog axes (using continuous jog).....	184
SINGLE BLOCK On/Off Switch	170	To Jog Axes (using incremental jog).....	184
BLOCK DELETE On/Off Switch (also called		To Use the Hand-Wheel	184
optional block skip).....	170	To Set Axis Displays	185
OPTIONAL STOP On/Off Switch	170	To Enter Tool Offsets (GEOM(H), WEAR(H),	
Buttons and Switches for Manual Functions	171	GEOM(D), WEAR(D)	185
Axis Jogging Controls	171	To Enter Workpiece Coordinate System Offsets	
Hand-Wheel Controls.....	171	185
Spindle Control	171	MDI Procedures	186
Automatic Tool Changer Control	171	To Execute an MDI Command.....	186
Indicator Lights and Meters.....	172	Program Manipulation Procedures.....	186
Spindle RPM and Horsepower Meters.....	172	To Get Ready to Edit Programs	186
Axis Drive-Motor Horsepower Meter	172	To Show a Directory of Programs	186
Cycle Indicator Lights	172	To Call Up a Program from Within the CNC	
Reference Return Position Indicator Lights		Memory (make it the active/main program)	186
.....	172	To Load a Program	187
Optional Stop Indicator Light.....	172	To Delete a Program	187
Other Buttons and Switches on the Machine		To Search Within a Program	187
Panel	172	To Alter, Insert, & Delete	187
Other Operation Panels on Your Machining		Alter	187
Center	172	Insert	187
More About the Mode Switch	173	Delete.....	188
The Manual Mode Switch Positions.....	173	To Save Programs to an External Device.....	188
The Manual Data Input (MDI) Mode Switch		To Use Background Edit	188
Position	174	Setup Procedures	188
Commanding an MDI Reference Return.....	174	To Mount the Work Holding Device on the	
The Procedure to Give an MDI Command ...	174	Table.....	188
Commanding a Tool Change with MDI	175	To Measure the Program Zero Assignment	
Commanding Spindle Activation with MDI..	175	Values in the X and Y Axes	189
Other Times When MDI is Used.....	175	To Measure the Program Zero Assignment	
Can You Make Motion Commands with MDI?		Value in the Z-Axis.....	189
.....	175	To Measure and Enter Tool Length	
The Edit Mode Switch Position	175	Compensation Values	189
To Make a Program in Memory the Active		Program Running Procedures	190
Program (to call up a program).....	176	To Run the Program (in normal production - no	
To Enter a New Program.....	176	verification techniques)	190
What if I make a mistake when typing?...	177	To Rerun Tools	190
The Program Operation Mode.....	177	To Do a Free-Flowing Dry Run.....	190
To Run the Active Program from the Beginning		To Do a Normal Air Cutting Run.....	190
.....	177	To Run the First Workpiece.....	191
Important Operation Procedures	181	To Cancel a Cycle.....	191
Develop Your Own Operation Handbooks	181	To Clear an Alarm.....	191
Manual Procedures:.....	181		

KNOW THE COMPENSATION TYPES	193		
Introduction To Compensation	195		
What is compensation and why is it needed?	195		
More On Tolerances	196		
The Initial Setting for Compensation.....	196		
When is Trial Machining Required?	196		
What Happens as Tools Begin to Wear?	197		
What do You Shoot For?	197		
Never make sizing adjustments by changing the CNC program!.....	197		
Understanding Offsets	198		
Offset Organization	198		
Offsets Related to Cutting Tools.....	198		
Offsets Related to Program Zero.....	200		
How Are Offsets Instated?	201		
Workpiece coordinate system offsets and Assigning Program Zero	203		
How is the program zero point determined?203			
What about program zero in the Z axis?	205		
Determining the Workpiece Coordinate System Offset Values	205		
Where is the Reference Position?	206		
How to Determine the Workpiece Coordinate System Offset Values.....	207		
How Predictable is the Work Holding Device?	207		
What is a Qualified Work Holding Setup?	207		
Machine, Workpiece, and Relative Coordinate Systems	207		
Manually Measuring Workpiece Coordinate System Offset Values.....	208		
Using an Edge-Finder and the Machine Coordinate System for XY with a rectangular workpiece.....	209		
Using an Edge-Finder and the Relative Coordinate System for XY with a Rectangular Workpiece	211		
Step 1: Load workpiece and edge-finder..	211		
Step 2: Touch off the workpiece edge in the X-axis	211		
Step 3: Set the relative position for the X-axis to zero	212		
Step 4: Back straight off in Z, move over 5.0 millimeters in the X-axis	212		
Step 5: Zero the X-axis relative position again	213		
Step 6: Touch off the workpiece edge in the Y-axis	213		
Step 7: Set the relative position for the Y-axis to zero	214		
Step 8: Back off in Z, move over -5.0 millimeters in the Y-axis	214		
		Step 9: Zero the Y-axis relative position again	215
		Step 10: Send the X- and Y-axes to the reference position	215
		Measuring the X- and Y-axes for a Circular- Shaped Workpiece	216
		Step 1: Load the workpiece and dial indicator	216
		Step 2: Indicate the center of the workpiece	216
		Measuring the Z-axis workpiece coordinate system offset value.....	217
		Step 1: Make the work holding setup and load a workpiece	217
		Step 2: Touch the spindle nose to the gauge block	217
		Step 3: Preset the Z-axis relative position to the height of the gauge block.....	218
		Step 4: Send the Z-axis to the reference position.....	218
		Measuring the Workpiece Coordinate System Offsets with a Spindle Probe	219
		Entering Program Zero Assignment Values into Workpiece coordinate system offsets	220
		Polarity of Work Offsets	221
		A Full Example of Using the Workpiece Coordinate System	221
		The Program	221
		The Setup.....	222
		Tool Length Compensation	227
		The reasons why tool length compensation is needed	227
		No Two Tools Will Have Exactly the Same Length	227
		Tool's Length Will Vary Each Time a Cutting Tool is Assembled.....	228
		Tool Data is Entered Separately from the Program	228
		Sizing and Trial Machining Must Often be Done	228
		What About Interference and Reach?	228
		The setup person's responsibilities with tool length compensation	228
		Which offset register is related to a given cutting tool?	229
		Recommended method: Using the tool's length as the tool length compensation value	229
		Determining tool length compensation values	230
		Measuring tool lengths on the machine.....	230
		Measuring Tool Lengths Offline—with a Tool Length Measuring Gauge	231
		Entering Tool Length Compensation Offsets	232

Using the Distance from the Tool Tip to Program Zero as the Tool Length Compensation (offset) Value	232	A Tip for Remembering Which Way to Adjust the Offset	251
Measuring tool length compensation values with this second method	233	How Important is it to Make Your First Workpiece a Good One?	252
Reasons for Using our Recommended Method	234	When Trial Machining Is Not Required	252
Trial Machining with Tool Length Compensation	234	Sizing with Cutter Radius Compensation	252
When Trial Machining is Not Required	235	What Do You Shoot for When Making Sizing Adjustments?	252
Sizing with Tool Length Compensation During a Production Run	235	What If I Use a CAM System and the Offset Value Is the Deviation from the Planned Cutter Size?	253
A Tip for Remembering Which Way to Adjust the Offset	235	Do I Have to Make All These Calculations When Adjusting Offsets?	253
The plus input key	236	Why Can't I Just Change Programmed Coordinate/s In the Program to Make Sizing Adjustments?	253
What If I Use the Second Method Shown for Tool Length Compensation?	237		
Why Can't I Just Change the Z-axis Coordinate/s In the Program to Make Sizing Adjustments?	237		
All Programmed Coordinates Must Specify Mean Values	237		
Programs Cannot Be Changed While the Machine Is Running	237		
		MASTER TASKS TO RUN A CNC MACHINING CENTER	257
Cutter Radius Compensation	243	Tasks Required to Setup a CNC Machining Center	259
Do you need to learn about cutter radius compensation?	243	Setup-related tasks	259
Similarities to tool length compensation	243	A CNC job from start to finish	259
Reasons why cutter radius compensation is required	244	Setup documentation	261
Coordinate calculations are simplified	244	Tear down the previous setup and put everything away	262
Range of cutter sizes can be used	245	Gather the components needed to make the setup	263
Do you use sharpened (re-ground) milling cutters?	246	Make the workholding setup	263
Trial machining and sizing	246	Squaring a vise	264
Rough and finish milling with the same set of coordinates	247	Assign the program zero point	264
The setup person's responsibilities with cutter radius compensation	248	Measure program zero assignment values ..	264
Which offsets are involved?	248	Enter program zero assignment values into workpiece coordinate system offset number one	266
The two ways to use cutter radius compensation	249	Assemble the cutting tools needed for the job	266
When the work surface is the programmed path	249	About the tightness of cutting tool components ..	270
When the cutters centerline path is programmed	249	Measure tool-length- and cutter-radius-compensation values	270
Relationship between the cutter radius compensation value and the machined surfaces	249	Reminder about the static nature of cutting tool measurements	270
What is the polarity for an adjustment?	250	Enter tool-length- and cutter-radius-compensation values into offsets	271
Trial machining with cutter radius compensation	250	Load cutting tools into the machine's automatic tool changer magazine	271
What If the Milling Cutter Machines on Both Sides of the Workpiece?	251	Warning about tool orientation	272
		Load the CNC program	273
		Physical tasks related to setup are now completed	273
		Verify the correctness of a new or modified program	273
		Verify the correctness of the setup	275
		We repeat a tip that will save a crash some day	275

Dry running our example program.....	275	Which takes longer, the CNC cycle or the off-	
What if you do find a problem?	276	line tasks an operator must perform?.....	303
Rerunning a tool in the program	277	Make offset adjustments to maintain size for	
When do you want the machine to stop?	279	critical dimensions (sizing)	303
Procedure to restart the program from the		Replace dull tools	304
beginning of a specific cutting tool	279	Clean the machine.....	305
Cautiously run the first workpiece	280	Preventive maintenance	305
The most dangerous time	280	Anything else you'll be expected to do?	305
The safe-approach procedure	280		
Making sure the first workpiece is a good one		ANSWERS TO EXERCISES	309
.....	281	Exercise 1: Basic Machining Practices	309
Machining the first workpiece in our		Exercise 2: What Does a CNC Machining Center	
example job	281	Do?	309
What if the program isn't perfect?.....	283	Exercise 3: General Flow of the CNC Process	309
If necessary, optimize the program for better		Exercise 4: Machine Configurations	310
efficiency	283	Exercise 5: Buttons and Switches on the	
If changes have been made to the program, save		Control Panels	310
the corrected version of the program.....	284	Exercise 6: The Key Operation Procedures	311
Verifying a Job that Contains Mistakes	285	Exercise 7: Introduction to Compensation	311
Loading the Program	289	Exercise 8: Workpiece coordinate system	
The Dry Run to Check for Setup Mistakes	290	offsets and Assigning Program Zero	311
Cautiously Running the First Workpiece	292	Exercise 9: Tool Length Compensation	312
		Exercise 10: Cutter Radius Compensation	313
Tasks Required to Complete a Production Run	297	Exercise 11: Tasks Required to Setup a CNC	
Production run documentation	298	Machining Center	313
Tasks required for completing a production run	299	Exercise 12: Tasks Related to Completing a	
		Production Run	314
Remove the previous workpiece.....	299		
Load the next workpiece	300	Answers to practice exercises: 273	
Activate the cycle	300	Index: 281	
Monitor the cycle	300		
Clean and de-burr the just-completed workpiece			
.....	301		
Perform specified measurements	302		