Course Outline

Mastering CNC Machining Centers

Course 1: The Basic CNC Machining Center

Lesson 1: The Computer Controlled Machine Tool

The introduction to this lesson identifies the brands and models of control that will be seen throughout the training program. The student will see the three most widely used brands, GE Fanuc, Mazak and Okuma CNC controls. Within each brand the student will see the models of control most commonly found in industry.

The lesson begins by identifying the basic operating principles of a CNC machining center. Both vertical and horizontal style machines are covered.

The mechanical components of the machining center are explained in the next section. The terms established here are used throughout the balance of the instruction when referring to these components. The lesson includes workpiece holding and loading mechanisms and several automatic tool handling and retrieval mechanisms.

Next, the lesson presents the basic components of the CNC control itself. The CRT, machine controls, soft keys, MDI keys, and so on, are explained. Within each component, a discussion covers the typical variations found on machines including such topics as color or monochrome CRT's, screen sizes, control panel configurations and so on. The basic soft-key menu systems is explored revealing part programs, offsets and position data displays. Lastly, the differences and similarities between the three brands of control are reviewed.

Lesson 2: Coordinate Grids on Machining Centers

As the most fundamental part of the CNC machining center and its operation, the coordinate grid is covered in detail in this lesson.

From a general discussion of the grid, the instruction moves into the specific application of the grid to the machining area of the machining center. Next, the units of measure used within the grid, and the axes of movement of the machine, are explained in detail. The rotational axes of movements are introduced as well. As many errors by operators involve mistaking the signs of coordinates and the signs of machine movements, the next section covers these concepts in detail. The idea and typical application of program zero and machine zero are introduced at this time.

Understanding the use of more than one grid system on a machining center has always been a point of confusion for new operators. The use of multiple zero points, and the resulting grids they establish, are explained in the next portion of the lesson. Further, the specific application of multiple grids as used on Fanuc, Mazak and Okuma models is covered.

Lesson 3: Machining Center Operations and Tooling

Now that the grid system has been explained, the instruction moves on to discuss the systems involved in positioning of tools within these grids.

The lesson begins with an explanation of the way in which the computer receives and interprets program commands to control the tool and work table movements. The speed of these movements is then broken down further into rapid traverse and feed rates. The concept of reducing cycle time is introduced as well.

The lesson then defines the basic machining movements needed to create a finished workpiece including such items as rough and finish cuts. The concepts of depth of cut, feed rate, and surface speed are then discussed.

The lesson then moves on to explain the specific types of machining movements the machining center can perform: milling, contouring, drilling, and so on. The types of milling cuts and the configuration of the cutters which perform these actions are explained. In addition, other machining operations such as reaming, boring, and tapping are explained, as well as the tools used to perform these operations. Because each machining process requires coolant supplied to the cutting tool, a discussion of the coolant system follows. The removal of the coolant and chips from the machining area is reviewed in the last portion.

Lesson 4: Automatic Machining and Manual Controls

With an understanding of the basic principles of machining center functions and tooling, the student is now ready to perform as a machine tender

This lesson prepares the student to take over an existing job and provides and maintain operations. It includes an explanation of the process of removing completed workpieces, cleaning fixtures, preparing the next workpiece, and locating and properly securing it in the fixture. The Cycle Start, Feed Hold, Emergency Stop, and Reset buttons are covered.

The equivalent manual machine controls are also explained. Comparisons between older- and newer-style control buttons and switches are included. A comparison is made between the names used to identify similar controls on each brand of control.

The controls used on Tool Changers, Pallet Changers and B axis rotary tables are also discussed. The operator then learns to jog the machining center components within each axis of travel using either the jog buttons or the Hand Pulse Wheel and its controls.

The final portion explains how to manually HOME the machine using the controls, and explains when this operation must be performed.

Course 2: Understanding Part Programs

Lesson 1: Programming Methods of CNC Machining Centers

Whether an operator ever needs to write a program or not, they will be required to read part programs so they can determine the corrective action needed to solve quality problems. This lesson introduces the most common languages used in machining center programming.

The lesson begins by explaining the basic concepts of computer programming languages and then carries the concept into the specifics of machining center programming. Three levels of program preparation are discussed: EIA, APT, and Conversational. Since APT and Conversational languages are normally translated into EIA codes before execution on the machine, a more detailed look at the elements of the EIA coding system is then provided.

Next, the lesson looks at the elements of a typical Conversational program as they appear on a machining center control. The Mazatrol language is used as the example.

In the last part of the lesson students learn about the capabilities added to the standard EIA/ISO programming language by various control manufacturers. As an example of these "language extensions", the basics of Okuma "User Task" programming is discussed. Topics include Common Variables and System Variables, conditional branching and unconditional branching using IF and GOTO, value passing to subprograms, math operators, and logic operators. All of these concepts are shown in the context of example part programs.

Lesson 2: EIA and User Task Programming Codes

The previous lesson in this series illustrated the importance of understanding EIA programming techniques, even on those machining centers which use Conversational programming methods. With this understanding, the student is ready to learn how specific EIA codes control machine movements.

The common G, M, S, T, and F codes are covered in the first three section. Since the process of suppressing leading zeros on some controls can make reading these codes within a program more difficult, this technique is explained. The division of the codes into modal and non-modal groups, and the default codes within each group, is covered in the following section. The concerns with non-linear G00 tool paths and the potential for collisions are also covered.

In the fourth part, Okuma User Task programming is explained in more detail. The goal of this instruction is to teach a machine operator to read and follow the logic of simple User Task commands and subprograms. All basic concepts are covered: variables and how they work, the four types of User Task variables, GOTO and IF commands, math operators, and logic operators.

Lesson 3: The Structure of EIA/ISO Part Programs

Now that the purpose of the individual codes within a program have been covered, an explanation of how these codes are organized into blocks and tool paths follows. This enables an operator to quickly find specific portions of the program when editing or troubleshooting is required. This lesson breaks a typical program down into initialization blocks, tool paths, reset blocks, and sub-programs.

Various Plane Selection codes, and G54 to G59 programming methods, are explained as it concerns editing and offset entry. This leads to a discussion of the correct terminology to use when reading or communicating both inch and metric dimensional values. The concepts of tenths and microns are covered. Next, absolute and incremental programming methods are explained as they apply to each control.

Lesson 4: The Structure of Conversational Part Programs

Similar to the previous lesson, this lesson covers the programming techniques used in the organization of Conversational part programs. It begins by discussing the basic steps in the writing of a conversational program.

Next, the elements of a Mazak conversational part program are explained. The student also receives an introduction to Mazak's Priority Machining function, since this influences the sequence in which the program is actually executed by the control. The last portion of the lesson explains the use of Sub Program, Pallet changing, Indexing, and End units.

Course 3: Machining Center Operator Skills

Lesson 1: Machine Start-up, Computer Memory, and Program Storage

Most CNC machinists begin their careers as machine operators. While the specific duties within shops may vary, generally this job classification includes machine start-up, checking operating systems, and activating a proven part program. This lesson provides the student with an explanation of these procedures.

Since part programs have been thoroughly covered in the previous course, this lesson explains how the various parts of a computer control's memory store and execute part programs.

The nomenclature used on each control to store files, programs and subprograms is covered next. The lesson then explains the process of displaying the Program Directory on each model. The data displayed in these directory screens is then reviewed in detail. The use of SEARCH functions and MDI keypad to locate stored programs names or numbers is also shown.

Lesson 2: Deleting, Loading, and Activating Part Programs

This lesson begins by explaining how various part-program file management systems are designed. It then discusses the capacity of the various storage mediums used including punched tape, floppy disk, and DNC.

Since the control's available memory may not be sufficient to hold a new program, the student is shown how to evaluate and select a program which can be deleted from memory, and to complete the deletion process on each model of control. Techniques for backing up programs are also explained.

Having learned how to make enough space available within the memory of the control, it is then possible for the operator to learn how to load both EIA and conversational programs. The process of loading a program begins with an explanation of the use of punched tape and tape readers. Next, the process of connecting and using the controls on a floppy disk drive are shown. Finally, the use of DNC systems, and the application of Front End Processor techniques for program storage are covered. The lesson also discusses the process of loading non-Mazak EIA programs into a Mazak M-Plus control and the precautions required before running them.

In the last portion of the lesson, the operator will learn how to make a program active on each brand of control, and how to detect the currently active program in each control.

Lesson 3: Understanding Tool Offsets

This lesson begins with a thorough explanation of the concept of tool offsets and how they are applied in both conversational- and EIA-based controllers. Also explained are the concepts of geometry and wear offsets for both tool length and cutter radius compensation. The data screens used for tool offset entry are shown for each brand and model of control covered in the program.

Since conversational controls require more comprehensive entry of tool and cutting conditions data, the Mazak Tool File and Tool Data screens, and the information they display, are explained.

The last section covers the use of Cutting Conditions and Material Selection data on a conversational control. The setting of feeds and speeds based on tool and workpiece material types is explained.

Lesson 4: Identifying Quality Problems

Statistical Process Control is one of the most common ways for operators to reduce scrap and rework. It allows the operator to monitor the process and make corrections before any out-of-tolerance parts are produced. While the extent to which SPC is applied will vary from shop to shop, this lesson discusses the basic principles behind SPC and the advantages it offers to the operator who uses it to eliminate the running of workpieces which do not meet quality standards.

The student's role as machinery operator will entail running a workpiece and checking and identifying any quality defects that occur. In addition, the learner must be prepared to identify quality defects as they appear in jobs for which he assumes operational responsibility. This lesson prepares the learner for these duties.

A large portion of the lesson deals with the thought process involved in troubleshooting, providing the student with the most efficient method of finding and eliminating quality problems, with an emphasis on reducing downtime. Also discussed are the causes of common quality defects and the most likely source of each problem. The quality problems covered include surface finish, location of features, and the size of features.

Lesson 5: Correcting Quality Problems and Program Restart

Having learned how to quickly identify the source of quality problems, the student is now instructed in making the adjustments necessary in order to resume production. This lesson first deals with stopping automatic execution once a defect has been found in order to make some of the more common corrections. The lesson goes on to teach the learner how to calculate a tool offset value, and determine its correct sign and address to overcome a quality defect. The incremental offset adjustment capabilities of each control are also discussed.

The next portion of the lesson enables the learner to identify chipped, burned, broken, and worn tools and cutting inserts through visual inspection. The process of removing, cleaning and replacing an insert is then explained.

The final segment of the lesson details the procedure for safely resuming operation. Also described are the Program Restart functions on all three controls, as well as Mazak's Tool Path Storage capabilities.

Course 4: Basic Setup and Programming Skills

Lesson 1: Process Planning, and the Selection and Installation of Workpiece Holding Devices

Because of the wider use of Conversational CNC controls, operators are often given the responsibility of writing simple workpiece programs. In addition, smaller shops which do not have off-line programming personnel or equipment may require the operator to assume all programming responsibilities on both EIA and conversational controls. This lesson prepares the student by giving him instruction in the task of planning the machining process.

The tasks include the reading and converting of dimensions on part prints to enable the writing of program coordinates, establishing a program zero location from the given print dimensions, and calculating rough-blank dimension requirements.

As an integral part of planning the machining processes that must be performed, the selection of work holding devices is then discussed. The lesson presents the different types of work holding devices that are normally found in use on machining centers, including modular fixturing systems. The student is shown the proper procedure for removing and installing fixtures as well.

Both EIA and conversational programs may require pallet changing and indexing. The final portion of this lesson provides the student with information on how EIA programs and conversational units control these functions.

Lesson 2: Tool Selection, Installation and Priority Machining Functions

With the work holding device selected and installed, the remainder of the set-up of the machine can begin.

This lesson continues the process by inspecting the existing tools which will be used for the new job, comparing them to the Tool Layout, and installing any new tools required. The student is given instructions on how to select the appropriate tooling for a new job. The types of cutting inserts, drills, boring bars, and the ANSI numbering system for tool inserts are also covered.

The lesson also explains the use of the Priority Machining function of conversational programs and its ability to reduce cycle times.

Lesson 3: Establishing Program Zero with the G92 Code

Several methods have been used to establish program zero based on programmer preference, and the age and capability of the CNC control. This lesson explains how G92 code is used in part programs, and how to find and enter the coordinate data when that code is used.

Part one of the lesson explains how to the G28 codes work in conjunction with the G92 code during program execution. It then explains how to use the Relative Coordinate Grid on Fanuc controls to locate Program Zero on a fixture or workpiece. It then explains how to enter these coordinate values on GE Fanuc 10M to 15M series controls. Part two covers the process of entering the G92 coordinate data on GE Fanuc 0M , 16M and 18M series controls.

In the third part of the lesson, the process of using and entering a G92 code data on a EIA-based Mazak control is covered.

Lesson 4: Okuma, Mazak, and G54 to G59 Program Zero Techniques

This lesson explains how the G54 to G59 code series is used to establish program zero on EIA-based machines. This opening section also includes the use of multiple zero points when multiple workpieces are machined by use of Local coordinate systems on both Fanuc and Mazak EIA-based machines.

Part two of the lesson explains how the G15 and G30 codes are used to establish the program zero and reference point locations on Okuma controls. The G11 Local Coordinate system code is also covered when multiple workpieces are involved.

The last part of the lesson explains how to locate and enter program zero on a conversational control like the Mazak. This section also details the use of Mazak's MMS unit for locating program zero from within the part program itself or by way of the MDI panel.

Lesson 5: Entering Tool Length Offsets

The method of establishing tool length offsets will vary in shops based on the techniques used to establish program zero and whether preset or non-preset tools are used. This lesson begins by explaining how these factors relate and their impact on the method of entering length offsets.

The next portion of the lesson covers the method used to find and enter the length offsets when a G92 code is used with non-preset tools. The following part covers the two most common methods of entering tool length offsets when G54 to G59 codes are used. These methods are determined by the technique used in the shop to establish the location of program zero.

Part four of the lesson covers entering tool length offsets on Okuma controls.

Part five explains how the tool-length plunger is used on Mazak controls to measure and enter length offsets. The lesson also covers automatic tool-length measurement techniques, and manual methods used on machines without tool-probes.

Lesson 6: Cutter Radius Compensation

Part one of the lesson explains the two methods that may be used by the programmer to apply Cutter Radius Compensation to tools. In part two, the student learns how the tool path is affected by a CRC offset. It includes learning when and how CRC is activated within a tool path, and the importance of the Plane Select codes in this process.

Part 3 explains how to locate and display the offset table, read the values entered to determine the technique being used, calculate CRC values when necessary, and enter them into the table. Part 4 explains how to correct for various feature dimensions by adjusting the CRC values. The variations found on some Fanuc controls are then revealed.

Part 5 covers the use of CRC on Mazak controls and the variations found in EIA and conversational program execution.

Course 5: Advanced Setup Skills

Lesson 1: Circular Interpolation and Manual Program Units

Whether an operator writes programs or not, they will be required to understand and potentially edit tool paths that include circular interpolation codes.

The first and second parts of this lesson detail the concerns associated with circular interpolation blocks within EIA programs and explains how to read and understand the tool paths created by various programming methods. To aid in correcting quality problems associated with circular interpolation blocks, an explanation is provided to enable the operator to identify which types of programmed blocks can be safely edited and which require action by a programmer.

Part 4 explains how and when EIA codes can be used within a conversational part program. The Mazak Manual Programming Unit is covered as the example.

Lesson 2: Drilling, Reaming, Tapping, and Boring Canned Cycles

Since canned cycles can reduce program size and simplify the programming process, part 1 of the lesson explains the basic programming and tool movements found in all canned cycles. It reveals how they are activated and deactivated. The variations found in canned cycles between each brand of control are also explained.

In part 2, a detailed look is taken at the various drilling canned cycles including G81, G82, G83 and G73. In part 3 the tapping cycles G84 and G74 are discussed in detail.

Part 4 covers the boring cycles G76, and G85 to G89.

Part 5 of the lesson explains what can be safely edited within canned cycle blocks to correct common quality problems.

Lesson 3 Conversational Programming Units for Drilling, Tapping, Boring, and Reaming

Conversational programming has the equivalent to canned cycles for each of the typical machining functions. The functions covered in this lesson are called Point Units in a Mazak conversational language.

Part 1 of this lesson explains Drilling Units. It includes a detailed look at the Unit, Sequence and Figure data blocks. It also explains how the size of the hole being specified will affect the programming displayed and how multiple-hole patterns can be programmed.

Part 2 covers the data associated with Tapping Units. Tapping of multi-hole patterns is also covered. Part 3 covers the four Boring Units while part 4 covers the programming found in Reaming Units and the tooling involved.

Lesson 4: Programming for Linear, Face, and Area Milling

Conversational Units are used to specify the milling process for linear and face machining.

Part one of this lesson covers the Line Unit in a Mazatrol conversational program. The instruction covers the data found in the Unit, Sequence, and Figure blocks for inside, outside, and chamfer cuts.

Part two covers the Face milling units found in Mazatrol programs. It includes an explanation of the values that must be entered in the Unit, Sequence, and Figure blocks for the top, step, pocket and slot programming units.

In part three, the Area Machining function found on Okuma EIA-based controls is covered. This includes the two Face cycles, two Round or Step milling processes, and two processes for Pocket milling.

Lesson 5: Verifying and Trial Running a New Program

Whether an operator is writing programs, or running programs produced by an off-line programmer, it is important to learn the steps in safely machining the first piece from a program which has not been run before.

In the first part of the lesson, the graphic simulation functions found on a conversational control like the Mazak are explained. The functions include Path Check, Trace, and Shape Check.

In part two, the use of Dry Run, Single Block, and Machine Lock is explained for all of the control types.

In part three, the actual trial run is completed. It includes the use of the Override controls to slow machine movements, the setting of the coolant lines to their proper positions, and the stopping of program execution to check features after each tool path is completed.

Lesson 6: Calculating and Entering Program Edits

Once the trial run is completed, the operator may find that some out-of-tolerance features may not be able to be corrected by using tool offsets. In these instances, it will be necessary to calculate and edit new coordinates into the program.

The first part of this lesson teaches the operator how to gather the critical information needed to determine if a coordinate should be adjusted to correct the problem. The operator then learns what information he must have to correctly make a coordinate adjustment. Since coordinates and correction values will have either plus or minus signs, the operator learns how to calculate a new coordinate using signed numbers. A further discussion covers the editing of Cutter Radius Compensation values to adjust for replacement cutters which may be undersize.

The second portion of the lesson explains all of the editing functions found on each brand of control and how they are used. This includes Insert, Alter, and Delete. The use of the Search and Memory Protect features are also explained.

NOTE: 25 Hours of Hands-On Machine Training Included.