

Course Outline

Mastering CNC Lathes

Course 1: The Basic CNC Lathe.

Lesson 1: Lathe Components and Principles of Operation.

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, Fanuc, Mazak and Okuma. 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 lathe.

Next, the types of material typically machined on a lathe are covered.

The mechanical components of the lathe are explained in the next section. The terms established here are used throughout the balance of the instruction.

Because of the variety of turret styles or automatic tool handling mechanisms found on CNC lathes, several configurations are shown along with an explanation of how each operates.

Next, the lesson presents the basic components of the CNC control itself. Within each component, a discussion covers the typical variations found on machines including such topics as color or monochrome CRT's, screen sizes, panel configurations and so on. Lastly, the differences between the three brands of control are discussed.

Lesson 2: The Coordinate Grid System.

As the most fundamental part of the CNC lathe 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 lathe. Next, the units of measure used within the grid, and the axes of movement of the machine, are explained in detail. The C axis of machining is introduced as well.

As many machining errors by operators involve the signs of coordinates and the signs of machine movements, the next section covers these concepts in detail. The program zero concept is introduced at this time.

The use of more than one grid system in a lathe has always been a point of confusion for students. The use of multiple zero points, and the resulting grids they establish, are explained to eliminate this problem. Further, the specific application of multiple grids as used on Fanuc, Mazak and Okuma models are covered.

Lesson 3: Lathe Machining Operations and Tooling.

Now that the grid system has been established, we can move on to discuss the positioning of the turret within the grid.

The lesson begins with an explanation of the way in which the computer controls the turret's movement. The speed of turret movements are 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 now moves into the specific types of machining movements that the lathe can perform, turning, facing, drilling, and so on. 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 covered in the following portion. Last, C axis driven-tooling concepts and operations are explained.

Lesson 4: CNC Lathe Programming Methods.

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 lathe programming.

The lesson begins by explaining the basic concepts of computer programming and then carries the concept into the specifics of lathe 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 lathe control. The Mazatrol language is used as the example. The lesson then explains how the Fanuc, Mazak and Okuma controls handle the execution of both EIA and Conversational programs. The concepts of Off-line and On-line programming are discussed to acquaint the student with the potential situations they may face within a shop and how it would affect their responsibilities.

Course 2: Understanding Part Programs

Lesson 1: Programming Codes and Manual Controls.

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

The G, M, S, T, and F codes are introduced in the first section. Since the process of suppressing leading zeros on some controls can make reading these codes more difficult, this situation 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. Also explained are the concerns with nonlinear G00 tool paths and the potential for collisions they create.

Following the explanation of the use of computer codes to control machine movements, the equivalent manual machine controls are explained.

Comparisons between older and newer style controls are covered.

Lesson 2: Reading EIA/ISO Part Programs.

Understanding the organizational approach used within an EIA program enables an operator to quickly find specific portions of the program when editing or troubleshooting. This lesson breaks down a typical program into initialization blocks, tool paths and reset blocks.

Diameter and Radius programming methods are explained as it concerns editing and offset entry. This leads into a discussion of the terminology used when referring to both inch and metric dimensional values. The concepts of "tenths", as it relates to one-ten-thousandth of an inch and microns are covered.

Next, absolute and incremental programming methods are explained as they apply to each control. The use of U and W addresses is included. Sub-program call structures and their applications are also covered.

Memory limitations in older CNC controls led to the use of a G50 code at the beginning of each tool path, a confusing and potentially dangerous technique. These concerns are addressed in the last portion of this lesson.

Lesson 3: Reading Conversational Part Programs.

Similar to the previous lesson, this lesson covers the programming techniques used in Conversational programs. It begins by comparing the basic steps in the writing of a conversational program between the three brands of control.

Next, the elements of a Mazak conversational part program are explained. The last portion of the lesson explains the most common machining processes used within a Mazak program

Lesson 4: Fanuc and Okuma Conversational Languages.

Just as lesson 3 dealt with the Mazak conversation language, this lesson discusses the Fanuc and Okuma conversational programming languages.

Part one progresses through each page of the data input screens used on lathes which use the Fanuc Symbolic FAPT language. A demonstration workpiece is programmed.

Part two covers the same process for Okuma IGF screens and data entry. Again, a demonstration workpiece is programmed.

Course 3: Lathe Operator Skills

Lesson 1: Loading, Storing, and Activating Part Programs.

Since part programs have been thoroughly covered in the previous course, this lesson explains how programs are stored, loaded and deleted from memory. The use of program directories and the data displayed on directory screens is explained for each of the brands covered.

Following this, the process of making a stored program an active program is revealed for each control type.

Loading of programs using tape readers, disk drives and DNC systems is then covered. The method of calculating program size and comparing it to available memory space is explained, as well as how memory space is allocated for program names and data.

Lesson 2: Tool Offsets and Tool Data Entry.

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. These include geometry, wear, tool nose radius and tool orientation. The data screens used for tool offset entry are shown.

Since conversational controls require more comprehensive entry of tool and cutting conditions data, the Mazak Tool File and Tool Data screens and their data displays are explained. The process of acquiring this tool data from standard tool catalogs and machining handbooks is also covered.

The last section covers the entry of Cutting Conditions and Material Selection data on a conversational control. The process of entering new material types and speed and feed data is then explained.

Lesson 3: Machine Start Up, Production Machining, and Quality Control.

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, activating a proven part program, trial running a piece and identifying any quality defects that may appear.

In addition, the learner must be prepared to identify quality defects as they appear in jobs for which they assume operational responsibility. This lesson prepares the learner for these duties.

To eliminate the running of workpieces which do not meet quality standards, a large portion of the lesson deals with the causes of common quality defects and the most likely source of each problem. The quality problems covered include surface finish, location of features, size of features, and defects in angular or radial cuts.

Lesson 4: Controlling Quality with Offsets, Tool Replacement, and Program Restarting.

This lesson teaches the learner how to calculate a tool offset value, and determine its correct sign and address to overcome a quality defect. Both OD and ID features are covered. The process of adjusting offsets with either the ADD or SET functions is also covered for each model of control.

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

On occasion, a tool may break during a cut or become so worn that continued machining would produce a poor quality workpiece. This lesson provides information on how to stop an executing program after a defect has been found and resume operation using a Program Restart function.

Course 4: Basic Setup Skills

Lesson 1: Process Planning for New Jobs.

Because of the wider use of Conversational controls, operators are often given the responsibility to write 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 learner by giving them instruction in the task of process planning.

The tasks include reading and converting of dimensions on part prints to enable the writing of program coordinates, establishing a program zero location, calculating rough blank dimensions, selecting work holding devices, planning machining processes and their order, and determining tooling requirements.

Lesson 2: Work Holding Devices.

This lesson begins by relating the quality and safety concerns that involve work holding devices. It also explains the various types of ID and OD chucks normally found in use. The mounting, positioning and machining of chuck jaws is then explained. Setting of gripping pressure is also covered.

Collets are covered in the next portion. It includes removal, installation, and adjustment of collet systems.

Conversational controls often require the entry of data on the workpiece holding device before machining begins. The entry of Chuck Data on Mazak controls is used as an example of this process.

The last section covers three types of tailstock. Installation of live centers, and the adjustment of each type of tailstock is included.

Lesson 3: Installing New Tools and Entering Tool Life Data.

With the work holding device selected and installed, the remainder of the setup of the machine can begin. This lesson continues the process by inspecting the existing tools which will be used for the new job and installing any new tools required. The types of tool holders, cutting inserts, drills, boring bars, and the ANSI numbering system for inserts and tool holders are covered.

Part two explains the Tool Life management system and its ability to activate a new tool after a tool has been used for the prescribed time or number of machining passes.

Lesson 4: Establishing Program Zero.

Program zero can now be located on the workpiece and entered in the computer. The lesson describes the manual machining of the workpiece using a reference tool, and the measuring and calculating necessary to locate the zero point. The process of entering the program zero data for each brand of control is explained in the four parts of the lesson. The last section covers the use of the G50 code to establish program zero and how and why multiple G50's may appear in some programs.

Lesson 5: Entering Tool Offset Data and Establishing the Safe Index Point.

With Program Zero established, the tool offsets for each tool can be determined and entered. Several techniques for touching-off tools are explained as well as the use of automated gauging systems and preset tools. The variations found in each brand of control are then discussed. Tool Nose Radius offsets and Tool Tip numbers to establish the tool orientation are also entered for each tool. The techniques for establishing and entering a Safe Index location are covered in the following section. Several approaches are explained to accommodate the most common programming methods and the specific features of each brand of control.

Course 5: Advanced Setup Skills

Lesson 1: Understanding Advanced Programming Techniques.

Whether an operator writes programs or not, they will be required to understand and potentially edit tool paths. This will include circular interpolation and canned cycle blocks.

The first portion of this lesson explains in more detail the concerns associated with circular interpolation commands within EIA programs. An explanation of portions of these blocks which can be safely edited is then covered.

Part two explains how and when EIA codes are used within conversational programs. The Mazak manual programming process is then covered.

The next section covers the impact of changing either Cutting Conditions or the Workpiece Material selections on a Mazak conversational control.

The last section of the lesson explains the basic concepts behind canned cycles and how they are utilized within EIA programs.

Lesson 2: Drilling, Grooving and Boring Canned Cycles.

With a better understanding of the linear and circular interpolation established in the previous lesson, the learner can now advance to the more complex machining cycles. Both the EIA canned cycles and the conversational processes used for drilling, boring and grooving are explained in this lesson. The student will understand which portions of the cycles may be safely edited to overcome quality defects. For those individuals who will be writing programs, this lesson will provide a greater understanding of the application of these special cycles.

Lesson 3: Turning, Threading and Facing Canned Cycles.

As in the previous lesson, this lesson expands upon the more complex cycles used to rough and finish turn, thread, and face workpieces. In the five parts of this lesson, both EIA canned cycles and conversational processes are covered.

Lesson 4: Machining the First Piece from 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 running the first piece. The use of Machine Lock and Dry Run features of each control are explained. The concept of entering trial offsets to avoid scrapping the first piece is covered as well. For those controls with graphic interfaces, the use of simulations of tool paths is then explained.

In part four, the operator learns how to calculate with signed numbers to establish the proper axis, sign and value when the need to adjust a coordinate instead of an offset arises. The final portion explains how to use the Insert, Alter and Delete editing functions of an EIA control.

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