SPRINGFIELD TECHNICAL COMMUNITY COLLEGE

ACADEMIC AFFAIRS

Course Number: MECH 150 Department: Mechanical Engineering Tech.

Course Title: Fundamentals of CNC Semester: Spring Year: 1999

Objectives/Competencies

Course Objective	Competencies
1.Perform the proper start-up procedures for both the lathe and the mill.	1.Discuss the significance of lab safety. 2.Understand all the Function Keys & Other Keyboard Components. 3.Mount tools in the holders and into the machines. 4.Establish Origin Point (Part Zero) on the lathe and mill. 5.Establish Tool Length Offsets for the lathe and mill. 6.Input Radius Offsets for the lathe and mill into the control unit.
2.Discuss the Cartesian Coordinate System as applied to both the lathe and mill.	1.Understand the axis directions and how to apply that knowledge to programming both the CNC lathe and mill. 2.Use a set-up sheet with each change of axis plotted to assist in programming.
3.Create CNC programs using the proper programming syntax.	1.Develop a working knowledge of G-codes including, but not limited to, G0, G1, G2, G3, G40, G41, G42, G80, and G81.

Course Objective	Competencies
	2.Develop a working knowledge of M-codes including, but not limited to M03, M05, M06. 3.Write programs using commonly used letter addresses that are modal.
4.Discuss and create a subroutine.	1.Analyze a part to determine if a subroutine will be helpful for the manufacturing process of the part. 2.Discuss program syntax appropriate for subroutine creation. 3.Write a subroutine using the required syntax.
5.Use Canned Cycles in a milling program.	1.Perform Synchronous Tapping. 2.Perform Right-Hand and Tapping Cycle. 3.Perform Chip-Break Peck Drilling Cycle. 4.Perform Deep Hole Peck Drilling Cycle.
6.Perform contouring on both the lathe and mill.	1.Discuss the cutter gauge point. 2.Calculate required cutter locations. 3.Use Rapid Traverses (G0), Linear Cutting (G1), Circular Cutting (G2 and G3). 4.Calculate and program lead in and lead out.
7.Use Tool Radius Compensation on both the lathe and mill.	1.Discuss the benefits of using tool radius compensation. 2.Discuss how to use tool radius compensation within programs. 3.Write program syntax to turn on cutter comp. 4.Write program syntax to turn off cutter comp.

Course Objective	Competencies
8.Create Straight tapers on the lathe.	1.Discuss program syntax for tapering. 2.Use the Command Point with Xc and Zc with regard to tapers.
9.Perform Automatic Corner-Rounding and Automatic Chamfering on the lathe.	1.Discuss the benefits for Auto Corner-rounding/Auto Chamfering. 2.Discuss program syntax. 3.Create programs using Auto Corner-rounding and Auto Chamfering.
10.Integrate Multiple Repetitive Cycles into a lathe program.	1.Perform Chip-Break Rough Turning. 2.Perform Drilling with G74. 3.Define and use G75, Chip-Break Rough Facing.
11.Process projects using proper program preparation.	1.Use a set-up sheet with each change of axis plotted to assist in programming. 2.Selecting proper tooling for the job. 3.Use the correct tool geometry for the job. 4.Select the correct insert size & shape. 5.Understand how to select proper insert grade. 6.Determine the proper cutting depth, speeds & feeds for the material being machined.