5/1/14	DATE			Technology	DIVISION
X	REQUIRED COURSE				NEW COURSE
	ELECTIVE COURSE			X	REVISION

LAKE LAND COLLEGE
Course Information Form

**COURSE
NUMBER**   **CIM 060**   **TITLE**   **Computer Numerical Control Machining**

SEM CR HRS 3.0   **LT HRS** 2.0   **LAB HRS** 2.0   **SOE HRS** _____ **ECH** _____

COURSE PCS# _____ (Assigned by Administration)

Prerequisites: Introduction to Machining Procedures MTT050, CAD056

Catalog Description (40 Word Limit): Introduction to computer numerical controlled (CNC) machine tool operation, programming and processes. Manual and computer-assisted part programming with machine tool verification.

List the Major Course Segments (Units)	Lt Hrs	Lab Hrs
History and Terminology	2.0	
Cartesian Coordinate System	2.0	
Feed and Speed Calculations	2.0	2.0
Manual CNC Lathe Operation	2.0	3.0
Computer Operation	2.0	2.0
CNC Lathe Programming	6.0	8.0
Manual CNC Mill Operation	2.0	3.0
CNC Mill Programming	6.0	8.0
Math for CNC	4.0	2.0
CAD/CAM Operation	2.0	2.0

EVALUATION: **Quizzes** _____ **Exams** X   **Oral Pres** _____ **Papers** _____

Lab Work X   **Projects** _____ **Comp Final** X   **Other** _____

Textbook: **Title:** Introduction to Computer Numerical Control

Author: James Valentino, Joseph Goldenberg

Publisher: Prentice Hall

Edition   4th

Copyright Date: 2012

Major Course Segment	Hours	Learning Outcomes
		The student will learn:
Introduction	4.0	Explain history, terminology and the coordinate system related to CNC machining.
Feeds and Speeds	4.0	Calculate spindle speeds and feed rates.
Lathe Operation	4.0	Operate a CNC lathe in the manual mode.
Computer Operation	4.0	Use CNC control software to enter and simulate a CNC program.
Basic Lathe Programming	4.0	Setup a basic lathe program
Advanced Lathe Programming	4.0	Program the CNC Lathe using arcs and canned cycles
Threading	4.0	Program a CNC lathe to cut threads.
Lathe Tooling	4.0	Select lathe tooling and to program tool offsets.
Mill Operation	4.0	Operate a CNC mill in the manual mode.
Basic Mill Programming	4.0	Write a basic mill program.
Math for CNC	4.0	Use basic trigonometry to calculate tool path points.
Circular Interpolation	4.0	Calculate circular programming points.
Canned Cycles	4.0	Write a CNC program using canned cycles
Subroutines	4.0	Write CNC programs with subroutines
Machining Center	4.0	Establish reference points and tool offsets on an industrial machining center.
CAD/CAM Operation	4.0	The basics of CAD/CAM operation

Course Outcomes: At the successful completion of this course, the students will be able to:

- Calculate spindle speeds and cutting speeds for a CNC machine.
- Write and execute a program for a CNC lathe.
- Write and execute a program for a CNC mill.
- Calculate programming coordinates using mathematical methods.
- Setup tool and fixture offsets on a CNC machining center.