SYLLABUS - CADD 31

COURSE: CADD 31 --- “Introduction to CATIA V5”

UNITS: 2

HOURS: 8 PER WEEK --- 2 LECTURE. 6 LAB

The course is 8 weeks long.

TEXT: “CATIA V5 Workbook, Release 18” - Cozzens

DESCRIPTION: CADD 31 is a lecture/lab course which covers the basic skills and knowledge required to use the CATIA V5 software. Subjects covered in the course include:

- CATIA V5 Interface
- Use of the Sketcher
- Creating and Managing Constraints
- Creating and Managing Solid Models
- Drafting
- Assembly Modeling

GRADING:	**ACTIVITY**	**POINT VALUE**
	Workbook Lessons 3-8 (25 Points Each)	= 150
	2 Workbook Exercises for each Lesson (10 Points Each)	= 120
	6 Solid Models From Handouts (30 Points Each)	= 180
	4 Drawings From Solid Models (20 Points Each)	= 80
	3 Assembly Models From Handouts (90 Points Each)	= 270
	Tests:	
	Midterm Examination	= 100
	Final Examination	= 100
	Total Points	= 1000

EXTRA CREDIT:

Extra Solid Models	=	25 each
Donate Blood (1 Pint)	=	25 each
Vote in Public Election	=	25 each

MAXIMUM POSSIBLE EXTRA CREDIT IS 50 POINTS

BONUS POINTS: If you score higher on the Final Exam than you scored on the Midterm Exam, you will receive bonus points equal to the difference in scores.

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To receive an “A”, you must earn at least 900 points.

To receive a “B”, you must earn at least 800 points.

To receive a “C”, you must earn at least 700 points.

To receive a “D”, you must earn at least 600 points.

You will receive an “F” if you earn less than 600 points.

STUDENT LEARNING OUTCOMES:

Creating CATIA V5 Simple 3D Solid Models:

Given a fully dimensioned multi-view engineering drawing of a machined part, the student will be able to utilize the appropriate functions within the CATIA V5 software to construct a 3D solid model of the part.

Creating CATIA V5 Simple Engineering Drawings:

Given a 3D solid model of a simple machined part, the student will be able to utilize the appropriate functions within the CATIA software to create a fully dimensioned multi-view engineering drawing of the part.

Creating CATIA V5 Simple Assembly Models:

Given a set of 3D solid model s of the component parts of a simple assembly, the student will be able to utilize the appropriate functions within the CATIA software to create a fully constrained assembly model.

DISABILITY STATEMENT:

Students with disabilities who believe they may need accommodations in this class are encouraged to contact the Special Resources Center on campus as soon as possible to ensure such accommodations are implemented in a timely fashion. Please contact me privately to discuss your specific needs.