COURSE OUTLINE
Loc Nguyen, 2012

DEPARTMENT:	Professional Technical Education
CURRICULUM:	CAD / DESIGN Technology
COURSE TITLE:	Intermediate CATIA
COURSE NUMBER:	TDR 241
TYPE OF COURSE:	Technical Preparatory
COURSE LENGTH:	1 quarter
CREDIT HOURS:	4
LECTURE HOURS:	22
LAB HOURS:	44
CLASS SIZE:	20
PREREQUISITES:	TDR 228 CAD Sheet Metal and TDR 240 Into To CATIA or Instructor permission

COURSE DESCRIPTION:

This is an extension of **CATIA** (**C**omputer **A**ided **T**hree Dimensional **I**nteractive **A**pplication) fundamental course by expanding students skills and techniques to further utilize advanced features of the Computer Based Three Dimensional Modeling. A hands-on course where students produce the parts drawings and assemblies, Surfacing and Sheet metal Drafting.

STUDENT LEARNING OUTCOMES ADDRESSED:

1. Communication - Read and translate technical data relative to geometric spatial relationships into a graphical form easily understood by others with similar technical understanding.

2. Computation - Use basic mathematical operations as required defining geometrical spatial relationships.

3. Human Relations - Use social interactive skills to enhance learning through informal tutoring activities.

4. Critical Thinking and Problem Solving - Organize and evaluate technical data, as well as select and apply appropriate spatial relationship principles to determine problem solution.

STUDENT LEARNING OUTCOMES ADDRESSED: (cont.)

5. Technology - Select and use appropriate technological tools to create technical graphics.

6. Personal Responsibility - Take pride in own work

7. Information Literacy - Access & use information from variety of resources / data

GENERAL COURSE OBJECTIVES:
Upon completion of the course the student will be able to:

1. Understand the basic functionality of CATIA parametric solid-modeling design software.
2. Understand the terminology and the creation process for sheet metal part design.
3. Define and manage sheet metal part parameters.
4. Understand the GSD (Generative Shape Design) workbench.
5. Create curves or surfaces to improve imported wireframe geometry
6. Understand how to manage parts in the context of an Assembly.
7. Produce CATIA drawings Parts and assembly layouts

TOPICAL OUTLINE:

		APPROX. HOURS
I.	Course Overview	1
II.	Introduction to CATIA Interface	11
III.	Generative Sheet Metal Design - SMD	18
IV.	Introduction to Generative Shape Design	12
V.	Part Modeling & Assembly	24
	Total	66

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