MFGT 141 - CATIA I

Approval Date: 02/20/2014  Effective Term: Fall 2014

Department: MANUFACTURING TECHNOLOGY
Division: Career Technical Education
Units: 3.00
Grading Option: Letter Grade
Transferability: CSU Transferable
Course Is: AA/AS Degree
Repeatability: Not Repeatable

Contact Hours per Term:
- Lecture/Discussion: 36.00
- Lab: 54.00

Associate Degree GE Applicability: Elective

Recommended Class Size: 15
-Rationale: This class is taught at ADI. The room that has the CATIA software only holds 15 students.

Discipline/Minimum Qualifications:
Manufacturing Technology

Catalog Description:
Introduces solid modeling using CATIA software. Topics include sketching, part design modeling, wire frame and surface design, and assembly design.

Schedule Description:
Introduces solid modeling using CATIA software. Topics include sketching, part design modeling, wire frame and surface design, and assembly design.

Student Learning Outcome:
Lecture:
1. Assess the CATIA tools, workbenches, functions, and operations used to create and edit basic 2D and 3D part models, products, and drawings.

Lab:
1. Create basic 2D and 3D part models, products (assemblies), and drawings.

Course Objectives:
Lecture:
1. Explain the CATIA commands and how they are used to produce Parts, Products (Assemblies), and Drawings.
2. Identify CATIA sketch and part design features that are used to create and edit various 3D model part files.
3. Explain how engineering and manufacturing 2D drawings are created and edited using views of 3D parts.
4. Evaluate product files created from inserted parts and sub-assemblies using assembly design fundamentals.
5. Explain the wireframe and surface design functions and features and how they are applied to create 3D wireframe and surface models.

Lab:
1. Design, draw, and edit 3D parts, drawings, and products (assemblies) using the CATIA
Construct and edit part models utilizing the sketcher and part design workbenches.

2. Assemble and edit 2D drawing files from 3D part views utilizing the drafting workbench tools.

3. Create and edit product assembly design models utilizing the assembly design workbench tools.

4. Produce and edit 3D wireframe and surface models utilizing the wireframe and generative shape design workbenches.

Course Content Outline:

LECTURE:

1. CATIA File types and file structure
   A. Part Models
   B. Assembly Models
   C. Drawings

2. CATIA Graphics User Interface (GUI) and Screen Layout
   A. Toolbars, Top Menus, Specification Tree, Compass
   B. Tools/Options, Settings
   C. Display Properties and Options

3. Part Design Modeling: Sketch Workbench
   A. Creating, manipulating, and editing sketches and sketch entities
   B. Sketch constraints and dimensions
   C. Sketch profiles and construction geometry

4. Part Design Modeling: Sketch Based Features
   A. Extruded Features
      i. Pad and Pockets
   B. Revolved Features
      i. Shaft and Groove

5. Part Design Modeling: Transformation Features
   A. Mirror, Symmetry
   B. Rectangular and circular patterns
   C. Translate, rotate, scaling

6. Part Design Modeling: Dress-Up Features
   A. Fillet, chamfer and draft
   B. Hole feature

7. Part Design Modeling: Additional Sketch Based Features
   A. Rib and slot
   B. Multi-section solids, multi-section solids remove

8. Drafting Fundamentals
   A. Creation of 2D drawing files with views from 3D models and assemblies
   B. Standard, isometric, projected, section and detail drawing views
   C. Creation and editing of dimensions, notes, tolerances, GD&T; symbols and drawing callouts
   D. Drawing standard formats and sizes
   E. Adding and editing title blocks and drawing borders

9. Assembly Design Fundamentals
   A. Creation and editing of product (assembly) files in the Assembly Design Workbench
   B. Inserting, manipulating, and constraining part model and products (assemblies)

10. Wireframe and Surface Design
    A. Wireframe workbench tools
    B. Generative shape design workbench
    C. Wireframe/surfacing features
        i. Fill, thickness, join, offset, split, trim
LAB:
1. Using the CATIA GUI, create and save basic 3D solid parts models.
2. Creating a 3D solid model utilizing various sketch entities, tools, and sketch constraints.
3. Creating a 3D solid model using sketches and pad, pocket, shaft, and groove features.
4. Creating a 3D solid model using dress-up features.
5. Creating a 3D solid model utilizing transformation features.
6. Creating a drawing with basic views and dimensions using the drafting workbench.
7. Creating an assembly model (product) from several parts, adding assembly constraints.
8. Producing a complete drawing including 3D parts and products.
9. Implementing rib, slot, multi-section solid and multi-section remove sketch based features.
10. Creating a helix wireframe and a surface model.

Methods of Instruction:
Lab, Lecture: Discussion and Educational software.

Methods of Evaluation:
Exams/Tests/Quizzes
Problem Solving
Projects
Skill Demonstrations

Typical Assignments:
Reading:
Text Readings, Handouts, Drawings, Online Resources.

Writing, Problem Solving or Performance:
Lecture:
1. Explain the tools and functions found in the Parts Design Workbench and how they would be applied to drawings/parts models.
2. Describe how you would create a 2-D drawing as used for manufacturing a part from a previously modeled 3-D part in Drafting Workbench.

Lab:
1. Model a Boeing 737 airplane, measuring a scale plastic model using calipers, rulers and graph paper. Make each wing a part; make the fuselage a part, as well as the vertical and horizontal stabilizers and the engines. Then create an assembly with the engine(s) on the wing(s), and put that into the top assembly.
2. Create a 3-D model part from a given dimensioned mechanical drawing (provided) in Part Design Workbench.

Other:

**Required Materials**

**Examples:**

**Book 1**

- **Author:** Sham Tickoo
- **Title:** CATIA V5-6R2012 for Designers
- **Publication Date:** 2012
- **Publisher:** CADCIM Technologies
- **Edition:** 2012

**Course Preparation:**

- **Prerequisite(s):** None
- **Co-Requisite(s):** None
- **Recommended:** None