**ATLANTIC CAPE COMMUNITY COLLEGE**

**COURSE TITLE**

**AERG200- Engineering Design**

**COURSE DESCRIPTION**

Designed to teach students freehand pencil sketching and visualization skills that will be used throughout their academic years and their professional career. Students are introduced to CATIA, a computer-aided design program predominately used in the aerospace industry. Students will utilize server-based communication and collaboration tools to work in groups to complete a design project.

*Prerequisite:* CISM125

*Credits:* 3

**TEXTBOOK**

Plantenberg, K. (2011). An Introduction to CATIA V6 Release 2012 –A hands-on tutorial Approach. Schroff Development Corporation Mission, KS

ISBN:9781585036639

Craig, J,W., Craig, O.,B. (2008). Engineering Graphics Text and Workbook Series 1.2.

Schroff Development Corporation Mission, KS

ISBN:9781585031337

**INTENDED LEARNING OUTCOMES**

Upon completion of this course students should be able to:

* Create freehand engineering sketches that utilize fundamentally sound concepts and techniques
* Utilize the computer aided design program CATIA to create engineering graphics that demonstrate fundamentally sound engineering design concepts and techniques

**LEARNING GOALS/OBJECTIVES**

1. **Students will be introduced to freehand sketching**
   1. Identify, sketch, and explain the function of points, construction lines, object lines, and hidden lines
   2. Plot points on grid paper to aid in the creation of sketches and drawings
   3. Explain the concepts of technical sketching and drawing
   4. Sketch an isometric view of simple geometric solids.
   5. Explain how an oblique view of simple geometric solids differs from an isometric view
   6. Sketch one-point, two-point, and three-point perspectives of simple geometric solids
   7. Describe the concept of proportion as it relates to freehand sketching
   8. Sketch multiview drawings of simple geometric solids
   9. Determine the front view for a given object
2. **Students will be introduced to CATIA.**
   1. Identify elements of CATIA’s interface
   2. Manage files using CATIA
   3. Select objects
   4. Edit Objects
   5. View objects
   6. Identify lifecycle operations
   7. Define maturity states
   8. Identify user roles
   9. Utilize the VML navigator
   10. Identify elements of the mechanical design workbench
   11. Utilize the specification tree
   12. Utilize the robot
   13. Export files
   14. Import files
3. **Students will learn how to use Sketcher.**
   1. Identify Sketcher work modes
   2. Set profiles
   3. Set constraints
   4. Utilize sketch analysis
   5. Modify relimitations
   6. Create axes
   7. Create transformations
   8. Operate on 3D geometries and sketch planes
   9. Utilize points
   10. Snap to point
   11. Cut a part
   12. Utilize splines
   13. Create circles
   14. Create arcs
   15. Add text
   16. Create lines Define material properties
   17. Modify geometries
   18. Re-limit geometries
4. **Students will learn how to make designs using CATIA.**
   1. Utilize pads
   2. Create pockets
   3. Create holes
   4. Create slots
   5. Utilize shafts
   6. Utilize grooves
   7. Utilize drafts, fillets, and chamfers
   8. Utilize shell, thickness, and threads
   9. Identify transformation features
   10. Identify Boolean operators
   11. Create ribs and slots
   12. Create stiffeners
   13. Apply a shell
   14. Add thickness
   15. Create threads
   16. Create circular patterns
   17. Mirror parts
   18. Create rectangular patterns
5. **Students will learn assembly design fundamentals.**
   1. Identify assembly constraints and advanced commands
   2. Identify elements of the assembly design workbench
   3. Define product structure
   4. Utilize the insert menu
   5. Identify the engineering connection toolbar
   6. Take measurements
   7. Conduct a space analysis
   8. Render images
   9. Apply engineering connections
   10. Create a wheel sub-assembly
   11. Duplicate parts
   12. Set up a product
6. **Students will learn drafting fundamentals using CATIA.**
   1. Identify parts of the view toolbar
   2. Utilize the annotations toolbar
   3. Model a part
   4. Create a standard view
   5. Create a sectional view
   6. Create dimensions
7. **Students will learn generative surface design fundamentals, while learning how to create basic wireframe surfaces and use them to create solids.**
   1. Identify elements of the wireframe toolbar
   2. Identify elements of the surfaces toolbar
   3. Identify elements of the points toolbar
   4. Identify elements of the planes toolbar
   5. Identify elements of the project combine toolbar
   6. Identify elements of the curve offset toolbar
   7. Identify elements of the circle-conic toolbar
   8. Identify elements of the surfaces toolbar
   9. Identify elements of the operations toolbar
   10. Create a wireframe
   11. Create a surface

**COURSE EVALUATION**

Tutorials (30) 300 points

Homework (15) 300 points

Projects (3) 300 points

Final Project (1) 100 points

A= 900-1000

B=800-899

C=700-799

D=600-699

F=599-

**LEARNING ACTIVITIES**

*Projects – Projects are based on the course tutorials and will require both freehand and computer aided graphical tools*

*Homework - Homework assignment are based on lecture, demonstration and reading*

*Final Project – The final project for the class is a group project requiring students to demonstrate concepts and techniques learned in the class (more information will be given in class)*

*Tutorials – Tutorials can be found in the course texts*