MAE 656. Advanced Computer Aided Design
Summer II 2012

Instructor:	Dr. Xavier Martinez
Contact:	x.martinez *"at"* upc.edu
Web:	https://web.cimne.upc.edu/users/xmartinez/
Schedule:	Mo-Thursday, 10:30 – 12:05, ESB-G78
Office hours:	Mo-Thursday, 12:05 – 1:30pm

References:

O. C. Zienkiewicz, R. L. Taylor, J.Z. Zhu, *The Finite Element Method: Its Basis and Fundamentals*. Sixth Edition. Butterworth-Heinemann (2005). ISBN 978-0750663205

Kent Lawrence. *ANSYS Workbench Tutorial Release 13*. Cambridge SDC Publications (2011), ISBN 978-1585036714

Huei-Huang Lee. *Finite Element Simulations with ANSYS Workbench 13. Theory – Applications – Case Studies*. SDC Publications (2011), ISBN 978-1585036530

ANSYS User Manual

Prerequisites:

An introductory course in finite element method and CAD applications (MAE 456 or equivalent)

Course Objective:

This course can be understood as a complement to any introductory course to the finite element method. The main objective of the course is to examine the available tools existing in commercial simulation packages, and learn how to use them; emphasizing their virtues and drawbacks.

Nowadays the most common procedure to design structural components is with the help of numerical simulations using FEM packages. It is common in engineering degrees to have courses about the FEM method. These courses are necessary to know the basic theory that lies behind a FEM code, but are not sufficient to learn how to use the code itself. Assuming the basis that to excel in the use of a FEM package requires many hours working with it; this course will teach the student how to use the code, what steps should be taken to perform a numerical simulation of a structural component, what tools are required in the FEM code to perform this simulation, and how this simulation is used to design a structural element.

After completing the course, the student will be capable of performing the numerical simulation of most structural components with the appropriate formulation, elements, materials, mesh and boundary conditions. Besides knowing how to perform the simulation, the student will also learn how to present a CAD project.

Although there will be some theoretical classes, most of the course is based on working with the computer and learning how to use a FEM code using it. With this aim, most of the classes will be conducted in the computer room and will consist on solving different numerical examples. Most of the work will be done with ANSYS FEM package.

Outline

1. The Finite Element Method. Theoretical background, formulation and implementation.
    - 1.1. Overview of the Finite Element Method
    - 1.2. Available elements in a FEM code and assimilation of those elements to structural components
    - 1.3. Solution of truss elements
    - 1.4. Solution of beam elements
    - 1.5. Solution of 2D elements
2. ANSYS Workbench
    - 2.1. Description of ANSYS Workbench environment
    - 2.2. Definition of engineering data
    - 2.3. Geometry definition
    - 2.4. Definition of boundary conditions
    - 2.5. Meshing
    - 2.6. Solution options
3. Solution of Beams and Trusses
    - 3.1. Solution of beam structures
    - 3.2. Solution of truss structures
4. Solution of solid problems: 2D and 3D
    - 4.1. Introduction
    - 4.2. Mesh effects on solid simulations
    - 4.3. Solution of 3D structures
    - 4.4. Solution of 3D structures with different materials and orientations. Application to the ply drop-off test of composite laminates
5. Solution of shell problems
    - 5.1. Shell and membrane elements. Theoretical background
    - 5.2. Solution of shell problems
    - 5.3. Solution of shell structures coupled with beam elements

    5.4. Definition of shell elements from solid elements
    5.5. Solution of shell structures coupled with solid elements
    5.6. Application to composite materials. Element orientation
6. Preparing and presenting a CAD project
    6.1. Model definition and relevant information
    6.2. Presentation of the results
    6.3. Description of the final project

Grading:

All subjects described in this syllabus will have a project associated. These projects will be made by teams of two students. Each student will have also to develop individually a final project consisting in a report describing the numerical simulation of a structural component. All projects must be presented before the due date in order to pass the course.

There will be a total of four course projects and one final project. Each course project corresponds to 1.75 points of the final grade and the final project represents 3.0 points, of a total of 10 points.

Although help and discussions are encouraged among the class at large, various teams are forbidden to work together and/or present essentially similar work. Similarities in their work will be penalized.

Projects will be graded based on correctness and quality of reports and analysis. Correctness, neatness, completeness, organization and quality of the narrative are important criteria.

NO AUDIT STUDENTS ALLOWED, WITHOUT EXCEPTION.