FINITE ELEMENT ANALYSIS
ME 477/677
Fall 2007

Instructor: Dr. Alan R. Kallmeyer
Office: Dolve 114, 231-8835
Office Hours: MW 1:30 – 2:30 pm, T-Th 10:30 – 11:30 am
E-mail: alan.kallmeyer@ndsu.edu

Textbook: *Finite Element Analysis: Theory and Application with ANSYS, 3rd Ed.*, Saeed Moaveni, Pearson Prentice Hall, 2008.

Lecture/Lab: MW: 3:00 – 4:50 pm, Dolve 215 & 212

Course Description

This course serves as an introduction to the theory and application of the finite element method, with an emphasis on the *use* of the method. The course is divided into two parts: a discussion of the concepts and theory behind the FE method, and the use and application of the method using the commercial software package ANSYS. The theory and application will be presented concurrently throughout the semester. Topics will include 2D and 3D stress analysis (linear and nonlinear), thermal analysis, contact problems, modeling techniques, and critiquing the results.

Grading

Undergraduate		Graduate	
Homework Problems:	15%	Homework Problems:	15%
ANSYS Assignments:	40%	ANSYS Assignments:	40%
Design Project (ANSYS):	15%	Research Project (ANSYS):	15%
Midterm & Final Exams:	30%	Midterm & Final Exams:	30%

Homework Problems
Traditional homework problems will be assigned periodically from the textbook. These problems may include derivations and analytical problems requiring hand computations.

ANSYS Assignments
Roughly 4-5 projects will be assigned throughout the semester requiring the use of ANSYS. These projects are to be done individually. A brief report describing the objectives of the analysis, modeling techniques used, and results must be submitted for each project.

Group Design Project/Graduate Research Project
A group design project will also be required by the end of the semester. This project will be conceived by the group, must be of a “design” nature (no unique answer will exist), and will require the use of ANSYS (with more than one iteration). Groups of 3-4 people will work on the project. A final report must be submitted detailing the objectives, modeling techniques and assumptions, and results of the project. In place of a design project, graduate students are required to complete a research-oriented project that uses advanced capabilities of ANSYS.

Exams
Two exams (midterm and final) will be given. The exams cover the theoretical material, general modeling techniques, etc. as discussed in lectures and the textbook, but do not specifically cover the use of ANSYS.

Course Outcomes

1. Students must understand the basic formulations used in the development of the finite element method.
2. Students must be able to construct, analyze, and interpret the results of a finite element model using commercially available software (e.g., ANSYS).
3. Students must have an understanding of the different element types available, and select appropriate elements for a particular type of analysis.
4. Students must have an understanding of the methods by which a model can be constrained and loaded, and be able to apply loads and constraints to a finite element model that reasonably represent the loads/constraints applied to the physical structure.
5. Students must have an understanding of the sources of error in a finite element model (e.g., mesh discretization error, numerical error, etc.), and apply techniques to reduce the potential for error in modeling.
6. Students must be able to analyze two and three-dimensional structural problems using FE software.
7. Students must be able to analyze two and three-dimensional thermal problems using FE software.
8. Students must be able to use FE software as a tool in the iterative design process.

Affected Program Outcomes

- Graduates must have the ability to apply knowledge of mathematics and science to solve engineering problems.
- Graduates must have the ability to identify, formulate, and solve fundamental engineering problems.
- Graduates must have the ability to identify a suitable model for a physical system, and apply principles of mechanics, thermodynamics, fluid mechanics, and materials to obtain a solution that satisfies the constraints.
- Graduates must have the ability to design a system, component, or process to meet desired needs within realistic constraints such as economic, environmental, social, political, ethical, health and safety, manufacturability, and sustainability.
- Graduates must have the ability to use techniques, skills, and modern engineering tools necessary for engineering practice.
- Graduates must be able to analyze a physical system, and create and validate models based on engineering and mathematical principles that correctly represent such systems and reflect current industry practices.
- Graduates must have the ability to communicate effectively.
- Graduates must possess knowledge of contemporary issues.

All work in this course must be completed in a manner consistent with NDSU University Senate Policy, Section 335: Code of Academic Responsibility and Conduct (http://www.ndsu.nodak.edu/policy/335.htm)

Any students with disabilities or other special needs, who need special accommodations for this course, are invited to share those concerns with the instructor as soon as possible.

Course Outline (subject to change)

Topic	Reading
<u>Introduction to the Finite Element Method</u>	
Background, classification of methods, history, nodes & elements,	1.1 – 1.4
modeling fundamentals, computational steps	1.8 – 1.9
<u>Formulation of Finite Element Equations</u>	
Direct formulation, minimum potential energy formulation, weighted residuals methods, assembly of global stiffness matrix	1.5 – 1.7
<u>Review of Matrix Algebra</u>	
Basic operations, solution methods, eigenvalues and eigenvectors, implementation in Excel	Chap. 2
<u>Trusses: Bar or Spar Elements</u>	
Basic formulation, coordinate transformation, 2D trusses, 3D (space trusses)	3.1 – 3.3
ANSYS: Introduction and 2D Truss Example (Ex. 3.1)	3.4 – 3.6 Handout
<u>Beams and Frames</u>	
Simple plane beams, direct method and minimum potential energy formulation, 2D beam elements, 3D beam elements, loads and boundary conditions (BCs), stresses in beams	Chap. 4
ANSYS Examples: Simple Beam and 3D Beam Analysis	Handouts
<u>2D Stress Analysis Problems</u>	
Plane stress vs. plane strain, stress-strain relations, strain-displacement relations, element stiffness matrix, loads and BCs	10.2
Element types: triangular, quadrilateral	7.1 – 7.4
Isoparametric element formulations	5.5, 7.6, 10.3
Gauss quadrature	5.6, 7.7
Stress calculations and failure analysis	10.5
ANSYS Example: 2D (Plane) Stress Analysis	Handout
<u>3D Stress Analysis Problems</u>	
Stress-strain relations, strain-displacement relations, element types, modeling considerations, loads and BCs	Chap. 13
Axisymmetric elements	7.5, 10.4
ANSYS Examples: Axisymmetric Analysis and 3D Stress Analysis	Handouts
<u>Modeling Considerations</u>	
Substructures, symmetry, constraints, meshing, loads and BCs, connections, errors and accuracy	Notes
<u>Thermal Problems</u>	
Heat flow equations, thermal stiffness matrix, thermal loads	6.1
and BCs, steady-state vs. transient problems, modeling considerations	Chap. 9
ANSYS Example: Transient Thermal Analysis	Handout
<u>Nonlinear Problems (Structural)</u>	
Material vs. geometric nonlinearities, solution methods, ANSYS considerations, material behavior	Notes
ANSYS Examples: Elastic-Plastic Stress Analysis and Nonlinear Contact Analysis	Handouts