

ME309: FINITE ELEMENTS IN MECHANICAL DESIGN

Tuesday/Thursday - 9:30-10:45 AM

Winter 2008

Objectives: The main objectives of ME309 are to give students an understanding of the theory of finite element analysis, and a working knowledge of the appropriate application of the method to engineering problems.

Instructors: Ellen Kuhl
Durand 217, 724-8988
ekuhl@stanford.edu

Office Hours:
Wed 10-12am, & by appointment

Marc Levenston
Durand 233, 723-9464
levenston@stanford.edu

Office Hours:
Thurs 2-4pm, & by appointment

Course TAs: Addala K. Bhargav
bhargav@stanford.edu

Office Hours:
Terman 104, TBD

Namkeun Kim
kimnk@stanford.edu

Office Hours:
Terman 104, TBD

Text: Course notes (referred to as “Modules”) are available on CourseWork. These notes are not intended to be “stand-alone,” but rather to complement the lectures.

Three optional textbooks are on reserve in Terman Engineering Library:
Finite Element Modeling for Stress Analysis, R. Cook, (ISBN 0-471-10774-3).
Schaum’s Outlines: Finite Element Analysis, G. Buchanan, (ISBN 0-07-008714-8).
A First Course in the Finite Element Method, D. Logan, (ISBN 0-534-38517-6).

Website: See coursework.stanford.edu: Most of the material can be found in the Course Materials folder; modules, tutorials, homework assignments and solutions.

Tutorial: A tutorial session will be held Tuesday (January 15, 2008) 9:30-10:45am. This session will be held in Terman 104 (The Elaine Cluster). Additional self-paced tutorials are posted on the Coursework site.

ANSYS: Most of the class will be running ANSYS version 11 on linux workstations located in Terman 104, which is open to students 24 hrs/day. Students may use other commercially available codes if they wish, however, the other code(s) used should be noted on all homeworks, projects, etc. Also, ANSYS can be run remotely over the network using UNIX (but tends to be slow).

A size-limited student version of the software (more than adequate for this class and many basic research applications) called ANSYS ED that runs on Microsoft Windows is available for \$100 from ANSYS. If you are interested in purchasing a copy for personal use, contact Dr. Levenston for ordering information.

Grading:

Grades will be based on four “regular” homework assignments (50%), a course examination (30%), and a final “super-homework/project” (20%).

Homework: Homework is due at the beginning of class on the date indicated on the class syllabus and listed below:

Homework #1 due Thursday,	Jan 24 th
Homework #2 due Thursday,	Feb 7 th
Homework #3 due Tuesday,	Feb 19 th
Homework #4 due Thursday,	Feb 28 st
Homework #5 due Friday,	Mar 14 th

Midterm: The Midterm will be open book/notes, and is on Tuesday, March 4th.

ME309: Course Calendar

Wk	Date		Course Modules	Schaum's	Assignments Distribute	Due
1	01-08	Bar elements (stiffness approach) global coordinate system boundary conditions thermal stresses	Module 1* Tutorials #0, #1*	Chp. 1, 2		
	01-10	Bar elements continued Introduction to a finite element code			HW #1*	
2	01-15	Tutorials #0-#2: Intro. to Unix, Using bar and beam elements, Generating a mesh by hand, Automatic meshing	Tutorials #0-#2*			
	01-17	Top down and bottom up modeling approaches	Intro. to Ansys : Part I (Chapt. 6 & 7)*			
3	01-22	Beam Elements (stiffness approach)	Module 2*	Chp. 4		
	01-24	Using the weak form to generate the Stiffness matrix (a more general approach)	Module 3*	Chp. 5	HW#2 HW#5	(HW#1)
4	01-29	Plane problem introduction: CST Element				
	01-31	Modeling with ANSYS				
5	02-05	Plane problem introduction: Q4 element	Module 4*	Chp. 6		
	02-07	Isoparametric elements			HW#3 HW#4	(HW#2)
6	02-12	Stress Calculation, Mesh Accuracy, Error Estimation, Other elements	Module 5*	Chp. 3		
	02-14	Stress Calculation, Mesh Accuracy, Error Estimation, Other elements (continued)	Module 6*			
7	02-19	Thermal Analysis (steady state, transient, thermal-stress)	Module 8*			(HW#4)
	02-21	Thermal Analysis (steady state, transient, thermal-stress)				
8	02-26	Modeling Approaches, Modeling Errors, Constraint Equations, Validation	Module 9*	Remaining Chp. 7		
	02-28	Special issues in FE modeling, TBA				(HW#4)
9	03-04	Examination				
	03-06	Special issues in FE modeling, TBA				
10	03-11	Special issues in FE modeling, TBA				
	03-13	Special issues in FE modeling, TBA				
	03-14 (Fri)	Final reports for HW#5 are due at NOON, hand in to Addala Bhargav (location TBD)				(HW#5)

* All course materials are available on the class website; go to <http://coursework.stanford.edu>

ME309: Modules 1-9, Table of Contents

Module #1: Introduction and the Bar Element

- 1.1 Introductory information on ME309, including class syllabus
- 1.2 Overview of the Finite Element Method
- 1.3 Stiffness Matrix Formulation
- 1.4 Direct Method #1: bar element
- 1.5 Analysis procedure (written in terms of bar elements)
- 1.6 Conversion from local "s" coordinate system to Global x-y Coordinate system
- 1.7 Element Assembly to Global Structure: Qualitative Representation
- 1.8 Enforcing Displacement Boundary Conditions
- 1.9 Bar Element Examples
- 1.10 Direct Method #2 (an alternate interpretation of a bar element's stiffness matrix)
- 1.11 The Bar Element in Ansys (Link1 element)

Module #2: The Beam Element

- 2.1 Using this same approach to generate the beam element's stiffness matrix
- 2.2 The Need to convert to the Global Coordinate System
- 2.3 Summary of Modules 1 & 2
- 2.4 A sample structure
- 2.5 Element Library in Ansys (BEAM3 and BEAM4)

Module #3: The Galerkin Formulation

- 3.1 General Background on Galerkin Finite Element Formulation
- 3.2 Using the Galerkin Method to Determine the Stiffness Matrix for a Bar
- 3.3 Using the Galerkin Method to Determine the Stiffness Matrix for a Beam
- 3.4 Generalization of the Method
- 3.5 Application to Plane Problems (Chapter 3 in Cook)
- 3.6 Ansys Information on Plane Elements

Module #4: Isoparametric Elements

- 4.1 A quick recap on the Galerkin Method
- 4.2 Summary of Constant Strain Triangular Element
- 4.3 Summary of Bilinear Quadrilateral (Q4) Element
- 4.4 Summary of Element Developed so far
- 4.5 Other elements that can be formulated in a similar manner
- 4.6 Other elements that can NOT be formulated in a similar manner
- 4.7 Gauss Quadrature

Module #5: Stress Calculations

- 5.1 Bar Elements and how to find stresses once the DOF are found
- 5.2 Beam Elements and how to find stresses once the DOF are found
- 5.3 2D CST and how to find stresses once the DOF are found
- 5.4 Bilinear Quadrilateral (Q4) Element and stress extrapolation to corners
- 5.5 From Cook: Comparative Examples and An Application

Module #6: The Energy Norm and other elements

- 6.1A The Energy Norm: an error approximation technique for displacement based problems
- 6.1B The Energy Norm: information from Ansys
- 6.1C The Energy Norm: information from Cook
- 6.2 Modeling Potpourri
- 6.3¹ Information on Element Types from Ansys Manual

Module #7: Modal Analysis (Natural Frequencies)

- 7.1 Dynamic Analysis and FEA
- 7.2A Eigenvalue Solution
- 7.2B Eigenvalue Solution reading from Cook
- 7.3A Example #1
- 7.3B Example #2
- 7.4 Reduction (from Cook)
- 7.5A Example #1 revisited
- 7.5B Example #2 revisited

Module #8: Thermal Analysis

- 8.1 1-D Heat Conduction Problems
- 8.2 Comparison of 1-D and 2-D Thermal Elements
- 8.3 Heat transfer example
- 8.4 Transient Thermal Analysis
- 8.5 Notes from Cook on Thermal Analysis
- 8.6 Thermal Modeling Considerations and an application (Cook)
- 8.7 Non-linearities in thermal analysis

Module #9: Misc Notes

- 9.1 A few notes on constraint equations
- 9.2 Thinking through an analysis