EML 5526
FINITE ELEMENT ANALYSIS & APPLICATIONS

Instructor: Nam-Ho Kim, Raphael T. Haftka
Class hour: 12:50-1:40 PM, MWF
Class room: 102 NEB
Office: 210 MAE-A
Office hour: MWF 5th period (11:45 – 12:35)
Phone: 352-846-0665
E-mail: nkim@ufl.edu
http://www.mae.ufl.edu/nkim/eml5526/

SYLLABUS

- Teaching Assistants
  - Mr. Vijay Jagdale
    - Office: 235 NEB, Phone: 392-2524
    - Office hour: TTh 4th period (10:40 – 11:30), e-mail: vjagdale@ufl.edu
- Textbooks:
  - Concepts and Applications of Finite Element Analysis 4th Ed, by R. D. Cook, D. S. Malkus, M. E. Plesha, R. J. Witt, Wiley, 2002
    - Required.
- Software:
  - Projects and some HWs will require FE software Abaqus
  - Download and install from http://campus.3ds.com/simulia/freese
  - Must use your personal computer, not lab computers

OFFICE HOURS

Period	Mon	Tue	Wed	Thu	Fri
7:25 - 8:15					
8:30 - 9:20					
9:35 - 10:25					
10:40 - 11:30		TA Vijay		TA Vijay	
11:45 - 12:35	Office hour		Office hour		Office hour
12:50 - 1:40	EML5526		EML5526		EML5526
1:55 - 2:45					
3:00 - 3:50					
4:05 - 4:55					

Instructor: Nam Ho Kim, 210 MAEA, 846-0665, nkim@ufl.edu
Instructor: Raphael T. Haftka, 220 MAEA, 392-9595, haftka@ufl.edu
Vijay Jagdale: 235 NEB, 392-2524, vjagdale@ufl.edu

Class Website: http://www.mae.ufl.edu/nkim/eml5526.html

GRADES

- Homework
  - All assigned homeworks must be submitted before starting the due date class. Solution will be posted on the class website. No late homeworks will be accepted.
- Exams
  - Two, equally contributing examinations
  - Tentative schedules: Feb. 24 (Exam1), Apr. 21 (Exam2)
  - Quiz: There will be quizzes during the class
- Projects
  - Two projects in finite element analysis using Abaqus. Formal report is required. 10% penalty for late submission and no acceptance after one week.
- Grading
  - Exams (40%), Projects (40%), Homework+Quiz (20%)

4

COURSE SCHEDULES cont.

Date	Class	Reading Assignment
1/6 W	Introduction to finite element analysis	Chapter 1
1/8 F	1-D Bar element, assembly, applying BC	2.1, 2.2, 2.5, 2.7
1/11 M	Stress, strain, stiffness matrix, plane truss	2.4, 2.6
1/13 W	Space truss, sparsity, Mechanical load, stress	2.8, 2.9
1/15 F	Thermal strain, stress; modeling symmetry	2.10, 2.11
1/18 M	M. L. King Holiday, No class	
1/20 W	**Introduction to Abaqus**	
1/22 F	Beam theory	2.3, 4.1, 4.2
1/25 M	Potential energy	4.3, 4.4
1/27 W	Rayleigh-Ritz method, FE interpolation	4.5, 4.6
1/29 F	FE equation for beam, distributed load	4.8
2/1 M	Plane frame, convergence	4.9
2/3 W	**FE analysis of beam using Abaqus**	
2/5 F	CST, LST elements	3.1, 3.2, 3.3, 3.4, 3.5
2/8 M	Q4, Q8, Q9 elements	3.6, 3.7
2/10 W	Project 1 assignment	
2/12 F	Numerical integration	3.9

5

COURSE SCHEDULES cont.

Date	Class	Reading Assignment
2/15 M	Drilling DOF, incompatible modes, reduced integration	3.10, 3.11
2/17 W	Stress calculation	3.12
2/19 F	**FE analysis of stress concentration**	
2/22 M	Review of exam	
2/24 W	First In term exam	
2/26 F	Galerkin Method in one dimension	5.1,5.3
3/1 M	Galerkin Method in 2D and mixed formulations	5.5, 5.6.
3/3 W	Review of formulation techniques.	Project 1 due
3/5 F	Isoparametric elements	6.1,6.2
3/8 M	Spring break, no class	
3/10 W	Spring break, no class	
3/12 F	Spring break, no class	
3/15 M	Quadrature and Q8,Q9 elements	6.3, 6.4
3/17 W	Incompatible modes, and static condensation	6.6, 6.7
3/19 F	Stress calculations	6.10, 6.11
3/22 M	Validity of isoparameteric elements and patch test	6.11, 6.12.
3/24 W	Review of Chapter 6.	

6

COURSE SCHEDULES cont.

Date	Class	Reading Assignment
3/26 F	Isoparametric triangles and tetrahedral	7.1,7.2
3/29 M	Coordinate transformation	8.1, 8.2, 8.3
3/31 W	Connecting dissimilar elements and fracture mechanics	8.5, 8.7
4/2 F	Reanalysis.	8.9
4/5 M	Ill-conditioning and the condition number	9.1-9.3
4/7 W	Decay test, residual and convergence rate	9.4-9.6
4/9 F	Multi-mesh extrapolation	9.7
4/12 M	Mesh revision and gradient recovery	9.8, 9.9
4/14 W	Adaptive meshing	9.9, 9.11
4/16 F	Review of Chapter 9	
4/19 M	Review for second in-term exam.	
4/21 W	Second in-term exam.	

7

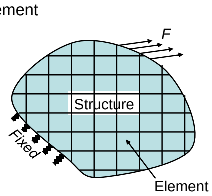
TIPS FOR A

- Be patient and persistent
  - Read the text repeatedly until you understand it.
  - If you don't understand it, ask a question until you get answered.
- Follow equations
  - Do not just read the equation.
  - You must follow all equations by **HAND**, **not EYE**.
- Try to understand the meaning of equations
  - If you memorize an equation that you don't understand, you can't solve the problem. Math is a language.
- Follow the instruction carefully
  - Read carefully what is asked. If A is asked, then answer A not B.
  - Do not submit a blank answer.

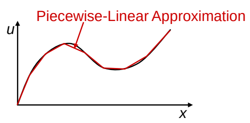
8

INTRODUCTION TO FINITE ELEMENT METHOD

- What is the finite element method (FEM)?
  - A technique for obtaining approximate solutions to boundary value problems.
  - Partition of the domain into a set of simple shapes (element)
  - Approximate the solution using piecewise polynomials within the element



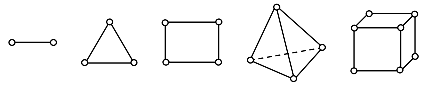
$$\begin{cases} \dfrac{\partial \sigma_{xx}}{\partial x} + \dfrac{\partial \tau_{xy}}{\partial y} + b_x = 0 \\ \dfrac{\partial \tau_{xy}}{\partial x} + \dfrac{\partial \sigma_{yy}}{\partial y} + b_y = 0 \end{cases}$$



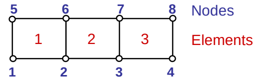
9

INTRODUCTION TO FEM *cont.*

- How to discretize the domain?
  - Using simple shapes (element)



  - All elements are connected using “nodes”.

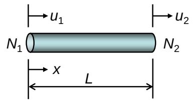


  - Solution at Element 1 is described using the values at Nodes 1, 2, 6, and 5 (Interpolation).
  - Elements 1 and 2 share the solution at Nodes 2 and 6.

10

INTRODUCTION TO FEM *cont.*

- Finite element analysis solves for Nodal Solutions.
  - All others can be calculated (or interpolated) from nodal solutions



  - Displacement within the element

$$u(x) = a + bx = u_1 + \frac{u_2 - u_1}{L}x = \frac{L-x}{L}u_1 + \frac{x}{L}u_2$$

Interpolation (Shape) Function

  - Strain of the element

$$\varepsilon(x) = \frac{\partial u}{\partial x} = -\frac{1}{L}u_1 + \frac{1}{L}u_2$$

11

INTRODUCTION TO FEM *cont.*

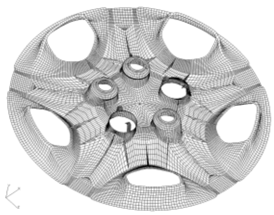
- How to calculate nodal solutions?
  - Construct a huge simultaneous system of equations and solve for nodal solutions.
  - Different physical problems have different matrices and vectors.

$$\begin{bmatrix} K_{11} & K_{12} & \cdots & K_{1n} \\ K_{21} & K_{22} & \cdots & K_{2n} \\ \vdots & \vdots & \ddots & \vdots \\ K_{n1} & K_{n2} & \cdots & K_{nn} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ \vdots \\ u_n \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ \vdots \\ F_n \end{Bmatrix}$$

12

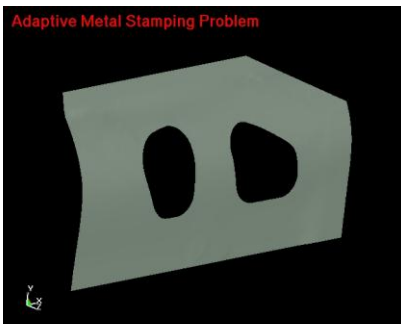
EXAMPLE: FINITE ELEMENTS

- Plastic Wheel Cover Model
  - 30,595 Nodes, 22,811 Elements
  - Matrix size is larger than 150,000×150,000.
  - MSC/PATRAN (Graphic user interface)



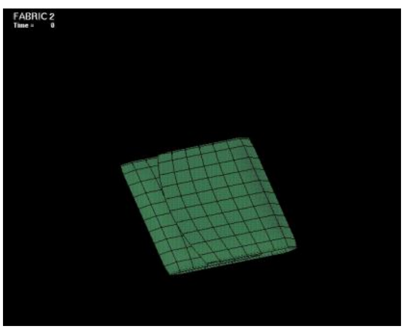
13

EXAMPLE: AUTOMOTIVE DOOR PANEL STAMPING



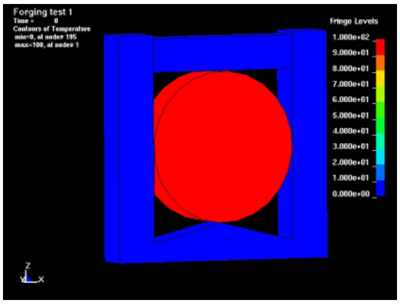
14

EXAMPLE: AIRBAG DEPLOYMENT



15

FORGING



16

EXAMPLE: VORTICITY

17