ME 455/555 *Intro. to Finite Element Modeling and Analysis*

Instructor: Hormoz Zareh, PhD, Associate Professor Emeritus
Mechanical and Materials Engineering Dept.

General information

This is a Remote "online only" class. Lectures will be on Mondays during the normal class time <u>and</u> will be synchronously broadcast via Zoom. You can either attend the lecture in person or view it remotely. In addition, there will be supplemental video sessions posted weekly. All lectures will be recorded and available for on-demand viewing via PSU Media Space. Virtual office hours will also be provided using the Zoom conferencing tool.

E-mail: hormoz@pdx.edu
Web: *https://web.cecs.pdx.edu/~hormoz/me455555/fall23/fea_outline_fall23.htm*
Remote Office Hours (via Zoom):
Mondays 9-10 am, Tuesdays 1-2 pm, Wednesdays 12:30-1:30 pm, Thursdays 1-2 pm

Grading Policy

Grades will be based on homework assignments and exams as follows:

Homework and computer projects	**20%**
Midterm exam	**35%***
Final exam and project	**45%**

* **No make up exam will be given. If you miss the midterm, final exam grade will be 80% of your total grade.**

Graduate Students (ME 555) project

Graduate students *<u>may</u>* need to complete an additional project (assignment) as part of the fulfillment of their grad-level coursework requirements.

The nature of the project will be discussed later in the term. Due date for completion will be on the day of the final exam.

Class policies

- Class attendance is not mandatory.
- All lectures will be recorded and available at the PSU Media Space website. Link will be emailed to the class.
- Please mute your microphone during lectures unless you want to make a comment or ask a question.
- Homework assignments are due at the specified date/time (uploaded to Canvas). Late homework will not be accepted. Graded HW will be posted to the Canvas site as well.
- Do not contact the grader with questions on HW assignments, Abaqus help, etc. You will be directed to me!

Course Objectives (Important concepts)

Understand fundamental concepts of Finite Element Techniques.

Will learn how to convert a physical problem description into an FE modeling problem.

Will learn how to set up the solution strategy necessary to obtain reasonable answer to the original physical problem.

[1] **Will learn how to use available computer simulation software to solve FE problems.**

Will learn how to interpret FE results and make sense of simulation findings.

[1] WILL NOT **confuse the course with commercial software vendors' "how to" training exercise.**

Course syllabus

Week 1: **Fundamental concepts, introduction to Finite Element Method**
Introduction to stiffness method

Week 2: **Development of Truss Equations**
Development of Truss Elements

Week 3: **Development of Beam Elements**
Frames and grids

Week 4: **Development of 2-D Plane Stress/Strain Equations**
Isoparametric Formulation

Course syllabus (continued)

Week 5: **Higher Order 2-D Elements**
Axisymmetric Elements

Week 6: **Midterm Exam (*tentative*)**
Three-Dimensional Solid elements

Week 7: **Intro. to Vibration (Structural Dynamics)**

Week 8: **Intro. to Heat Transfer problems , Thermal stress**

Select topics in Modeling, Pre- and Post- processing, Results interpretation, and Error analysis will be covered throughout the term.

FE software and access

Software: ABAQUS/CAE is available in the labs (EB 420, EB 325) and will be used for in-class demonstrations. However, you are free to use any available software of your choice (such as Ansys , Nastran (NX, MSC, …), Solidworks Simulation, even SAP2000) as long as it can handle the scope of topics and elements that are covered in the syllabus.

Access to the labs and software: You should be able to use "remote desktop" feature of Windows operating system to access the computers/software in EB 420. Please contact Computer Action Team (CAT) support for details.

Student version of ABAQUS is available as a free download (max. 1000 nodes) directly from CAT web site (need to log in with MCECS credentials). Be sure to download the installation instructions and carefully follow the notes.

https://cat.pdx.edu/services/software/users/abaqus/

The Fundamentals

What is Finite Element (FE) Method or Analysis?

A matrix solution technique initially developed by structural engineers at Stuttgart (Germany), Berkeley, and NASA. It was further developed by mathematical theories.

Trivia: In 1960 the term "Finite Element" was coined by Ray W. Clough of Berkeley.

Modeling: Dividing a structure or a continuum into a series of discrete, well-formulated entities (known as Elements).

Analysis: Approximation of field quantities within these discrete elements and combining the elements into an approximate behavior of the initial physical problem.

Two Direct approaches

Both of these methods can be applied to small problem sizes where formulation can be derived directly from the known set of governing equations.

- Force or flexibility method
  - Uses internal forces as unknowns and sets up a set of simultaneous equations based on equilibrium condition.
- Displacement or stiffness method

  Governing equations are based on the internal displacements. Here the condition of compatibility is satisfied.

Displacement Approach

- Based on assumed displacement field approximation
- Nodal displacement compatibility
- Governing equations of equilibrium
- Computationally efficient
- Widely used in general-purpose FEA codes
- Valid for structural analysis problems only!

Variational Method

- Valid for structural as well non-structural problems
- Based on the following principles:
  - Minimum potential energy applies to linear elastic material only
  - Virtual work can be applied to both linear as well as nonlinear materials
- Involves polynomial approximation of field quantity

Examples of Field quantities

Stress analysis:	Displacement field
Thermal Analysis:	Temperature or heat flux
Fluid Flow Analysis:	Velocity (potential) function

Mathematical explanation:

FE Method is a piecewise polynomial interpolation. It involves converting a governing set of complex partial differential equations into a much larger set of approximate simple algebraic equations.

Within each element, a field quantity such as displacement is interpolated (as a polynomial) from values at the nodes. By connecting the elements together, the field quantity becomes interpolated over the entire structure (or continuum) in a piecewise fashion.

The “best” approximation for the field quantity is the one which minimizes the deviation from “exact” solution. This deviation is usually measured by potential energy functions.

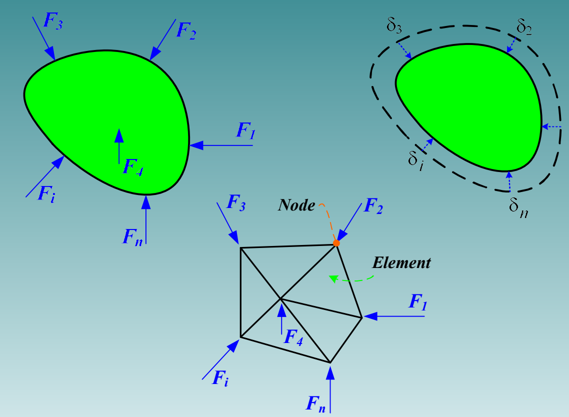
The displacement-based forms of these functions are called “strain-energy” functions.

Element Formulation Approach

Direct stiffness method
Potential Energy *
Weighted Residual (Galerkin)

The potential energy approach is equivalent to virtual work principle and can be stated as follows:

Assume a structure under application of forces is in a state of equilibrium. If the structure deforms due to a set of small “compatible” virtual displacements, the (virtual) work done by the external forces is equal to the (virtual) strain energy of the resulting internal stresses (or strains).



At the element level:

$\delta U_e = \delta W_e$

where

$\delta U_e \equiv$ Virtual strain energy

$\delta W_e \equiv$ Virtual work of external forces acting through virtual displacements

Finite Element solution involves selection of an "approximate" function to describe the displacement behavior within an element.

From displacement approximation, strain and stress fields are computed and the following expression is formed:

$\int_{Vol} \{\sigma\}^T \{\varepsilon\}\ d\ Vol =$ Work done by all external forces

where $\{\varepsilon\} = [\partial]\{u\}$ is the strain equation (per element) and $[\partial] = \begin{bmatrix} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \\ \frac{\partial}{\partial z} \end{bmatrix}$

Combining the total strain energies and work of all elements, and equating work with energy, results in the following general set of simultaneous equations:

$[K]\{U\} = \{F\}$

This equation can be solved to obtain the displacement values for all the points (nodes) which are vertices of the elements.

The obtained displacements can now be "back-substituted" into individual element's equilibrium equation to obtain element forces and, subsequently, stresses and strains.

$[K]\{u\} = \{F\}$ → will be solved to obtain $\{u\}$ for each element. From displacement values, strain $\{\varepsilon\}$ within each element is computed. For example, in a one-dimensional (Rod) element:

$$\varepsilon = \frac{u_2 - u_1}{L}$$

Finally, stress is determined by:

$\{\sigma\} = [E]\ \{\varepsilon\}$ → determines the stress per element.

Modeling the problem and checking the results

Modeling consists of simulation of a physical structure or a physical process by means of analytical or numerical constructs.

Therefore, it requires a good understanding of the problem that is to be modeled.

This understanding helps in determining the type of discretization suitable for a given problem, number of elements used in the model, approximation of physical boundary conditions, including loading and restraint, among others.

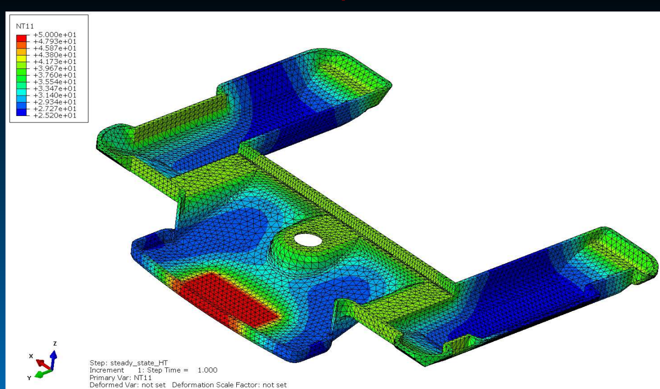
Discretization and other approximations

During FE study, we analyze a mathematical approximation of a physical problem. The approximation will invariably introduce *modeling error*.

As an example, the elementary beam theory ignores the transverse shear deformation. While this may be an excellent approximation for long slender beams, it is not valid for short non-slender ones.

If the beam to be modeled is deep, transverse shear deformation may become important and the element formulation should account for it. If the beam is very deep, then the elementary beam formulation may be altogether incorrect, and a two- or three-dimensional element formulation may be more appropriate.

An example



User Responsibility

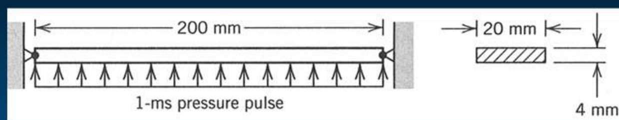
As the modern FE software has become easier to use, with colorful results contours, even inept users can produce some sort of an answer. This means that results can be produced by both good and bad models.

Flawed FE models including poor mesh, inappropriate element types, incorrect loads, and improper support restraints may produce answers that appear reasonable upon casual inspection.

Understanding the physical nature of the problem as well as the behavior of the finite elements are important prerequisites to prepare the user well enough to undertake an FE-based analysis.

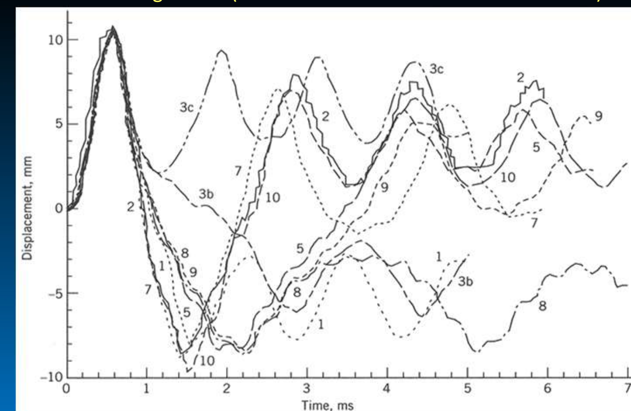
Example problem

Find the displacement response at the center of the beam



Reference: Concepts and Applications of Finite Element Analysis by Cook, Malkus, Plesha, Witt, 4th ed. ©2002 John-Wiley and Sons, Inc.

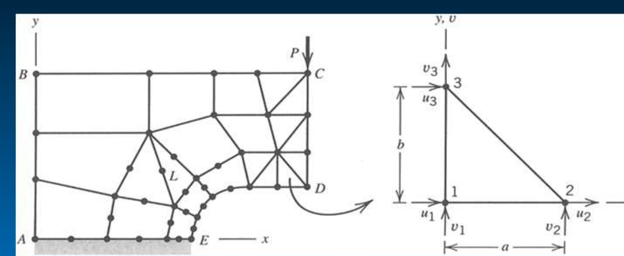
FE modeling results (Note variations based on different codes)



Basic building blocks in the Finite Element solution

1. **Discretization based on appropriate formulation**
2. **Application of material and physical properties**
3. **Assembly of elements**
4. **Load and Constraint application**
5. **Global solution**
6. **Results extraction and interpretation**

As an example, consider the general plane triangular element. The *x*-dir. and *y*-dir. displacement components of any given point inside the element are represented by *u* and *v* respectively (nodal displacements are indicated by corresponding subscripts).



Assuming a linear polynomial for approximation of displacement field:

$u = \beta_1 + \beta_2 x + \beta_3 y$

$v = \beta_4 + \beta_5 x + \beta_6 y$

Where β_i's are displacement field coefficients, also known as "*generalized coordinates*."

These coefficients are determined by substituting the coordinates (x and y) of the nodes and solving the resulting set of 6 simultaneous equations.

Boundary conditions:

$@\, x = 0$ and $y = 0 \rightarrow u = u_1$

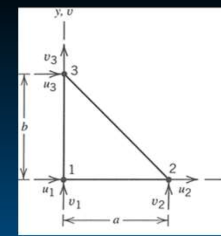
$@\, x = 0$ and $y = 0 \rightarrow v = v_1$

$@\, x = a$ and $y = 0 \rightarrow u = u_2$

$@\, x = a$ and $y = 0 \rightarrow v = v_2$

$@\, x = 0$ and $y = b \rightarrow u = u_3$

$@\, x = 0$ and $y = b \rightarrow v = v_3$



Substitute and solve to obtain:

$\beta_1 = u_1 \qquad \beta_2 = (u_2 - u_1) / a \qquad \beta_3 = (u_3 - u_1) / b$

$\beta_4 = v_1 \qquad \beta_5 = (v_2 - v_1) / a \qquad \beta_6 = (v_3 - v_1) / b$

Substituting for the coefficients in the polynomial equations:

$$u = (1 - \frac{x}{a} - \frac{y}{b})u_1 + \frac{x}{a}u_2 + \frac{y}{b}u_3 \qquad v = (1 - \frac{x}{a} - \frac{y}{b})v_1 + \frac{x}{a}v_2 + \frac{y}{b}v_3$$

Now we can obtain strains within the element by using the classical strain-displacement relations:

$$\varepsilon_x = \frac{\partial u}{\partial x} \quad or \quad \varepsilon_x = \beta_2$$

$$\varepsilon_y = \frac{\partial v}{\partial y} \quad or \quad \varepsilon_y = \beta_6$$

$$\gamma_{xy} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \quad or \quad \gamma_{xy} = \beta_3 + \beta_5$$

We'll note that the strain terms do not depend on either x or y. Thus, the proposed linear displacement approximation results in a state of constant strain within the element! The concept will be further discussed in 2-D element formulation.