Finite Element Analysis

Module 5 - CO4

Dr M B Vaghela
Assistant Professor
Mechanical Engineering Department
L. E. College Morbi
Email: mbvaghela@lecollege.ac.in

Objectives of This FEM Course

- ➢Understand the fundamental ideas of the FEM
- ➢Know the behavior and usage of each type of elements covered in this course
- ➢Be able to prepare a suitable FE model for given problems
- ➢Can interpret and evaluate the quality of the results (know the physics of the problems)
- ➢Be aware of the limitations of the FEM (don't misuse the FEM - a numerical tool)

Computer Implementations

- Preprocessing (build FE model, loads and constraints)
- FEA solver (assemble and solve the system of equations)
- Post processing (sort and display the results)

Available Commercial FEM Software Packages

- *ANSYS (General purpose, PC and workstations)*
- *SDRC/I-DEAS (Complete CAD/CAM/CAE package)*
- *NASTRAN (General purpose FEA on mainframes)*
- *ABAQUS (Nonlinear and dynamic analyses)*
- *COSMOS (General purpose FEA)*
- *ALGOR (PC and workstations)*
- *PATRAN (Pre/Post Processor)*
- *HyperMesh (Pre/Post Processor)*
- *Dyna-3D (Crash/impact analysis)*

Finite Element Analysis (FEA) or Finite Element Method (FEM)

- The Finite Element Analysis (FEA) is a numerical method for solving problems of engineering and mathematical physics.

- Useful for problems with complicated geometries, loadings, and material properties where analytical solutions can not be obtained.

A Brief History of the FEM

- 1943 ----- Courant (Variational methods)
- 1956 ----- Turner, Clough, Martin and Topp (Stiffness)
- 1960 ----- Clough (“Finite Element”, plane problems)
- 1970s ----- Applications on mainframe computers
- 1980s ----- Microcomputers, pre- and postprocessors
- 1990s ----- Analysis of large structural systems

FEM in Structural Analysis

Procedures:

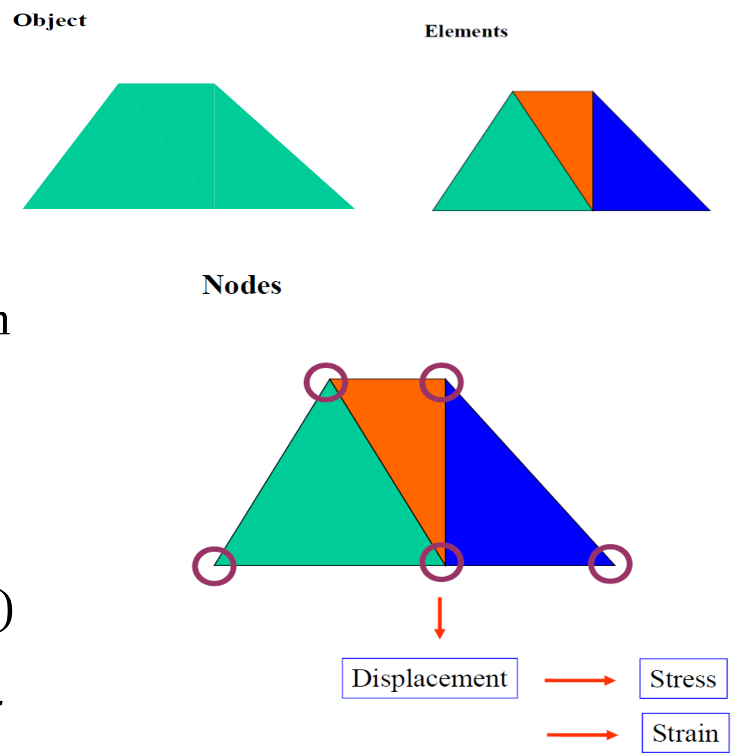
✓Divide structure into pieces (elements with nodes)

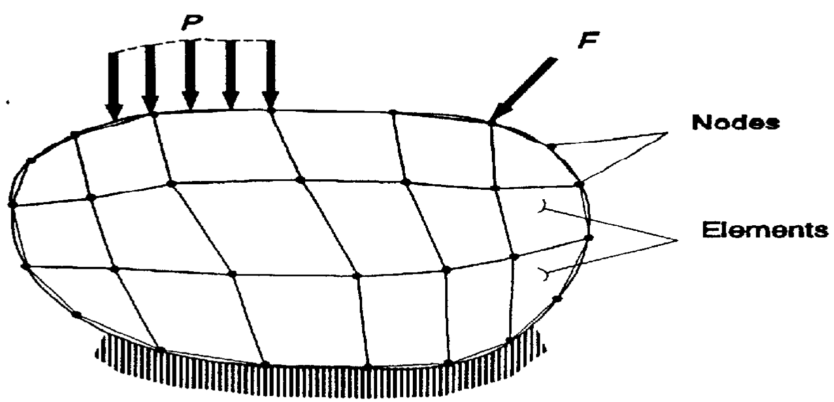
✓Describe the behavior of the physical quantities on each element

✓Connect (assemble) the elements at the nodes to form an approximate system of equations for the whole structure

✓Solve the system of equations involving unknown quantities at the nodes (e.g., displacements)

✓Calculate desired quantities (e.g., strains and stresses) at selected elements





WHAT IS FINITE ELEMENT ANALYSIS?

- Finite Element Analysis (FEA) is a method for numerical solution of field problems.

Examples for field problems

The distribution of temperature in a turbine blade

The distribution of displacements and stresses in an helicopter rotor blade.

- A field problem is described by differential equations or by an integral expression and either description may be used to formulate finite elements.

WHAT DOES “STRUCTURE” MEAN ?

- Here the word “structures” implies any solids that are subjected to loads or other influences.

- Such influences cause deformations (or strains) through out the continuum, accompanied by internal stresses and reactions at restrained points.

- The primary objectives of analysis by finite elements are to calculate approximately the stresses and deflections in a structure.

HOW DOES THE FEM ORIJINATE ?

- The finite element method is originally developed to study the stresses in complex aircraft structures.
- Then, it is applied to other fields of continuum mechanics, such as heat transfer, fluid mechanics, electromagnetics, geomechanics, biomechanics.
- However, this course is devoted solely to the topic of finite elements for the analysis of structures.

WHY FEM IS NECESSARY ?

- Analytical solutions to the engineering problems are possible only if the geometry, loading and boundary conditions of the problem are simple.

- Otherwise it is necessary to use an approximate numerical solution such as **FEM**.

WHY FEA IS USED IN INDUSTRY?

- To reduce the amount of prototype testing
  - Computer simulation allows multiple "what-if" scenarios to be tested quickly and effectively.
- To simulate designs that are not suitable for prototype testing
  - Example: Surgical implants, such as an artificial knee
- The bottom line:
  - Cost savings
  - Time savings… reduce time to market!
  - Create more reliable, better-quality designs

The Purpose of FEA

Analytical Solution

- Stress analysis for trusses, beams, and other simple structures are carried out based on dramatic simplification and idealization:
  - mass concentrated at the center of gravity
  - beam simplified as a line segment (same cross-section)
- Design is based on the calculation results of the idealized structure & a large *safety factor* (1.5-3) given by experience.

FEA

- Design geometry is a lot more complex; and the accuracy requirement is a lot higher. We need
  - To understand the physical behaviors of a complex object (strength, heat transfer capability, fluid flow, etc.)
  - To predict the performance and behavior of the design; to calculate the safety margin; and to identify the weakness of the design accurately; and
  - To identify the optimal design with confidence

APPLICATION AREAS

- FEA software packages are used by engineers worldwide in virtually all fields of engineering:
  - Structural
  - Thermal
  - Fluid (CFD, Acoustics, and other fluid analyses)
  - Low- and High-Frequency Electromagnetics
- A partial list of industries in which FEA is used:
  - Aerospace
  - Automotive
  - Biomedical
  - Bridges & Buildings
  - Electronics & Appliances
  - Heavy Equipment & Machinery
  - MEMS
  - Sporting Goods

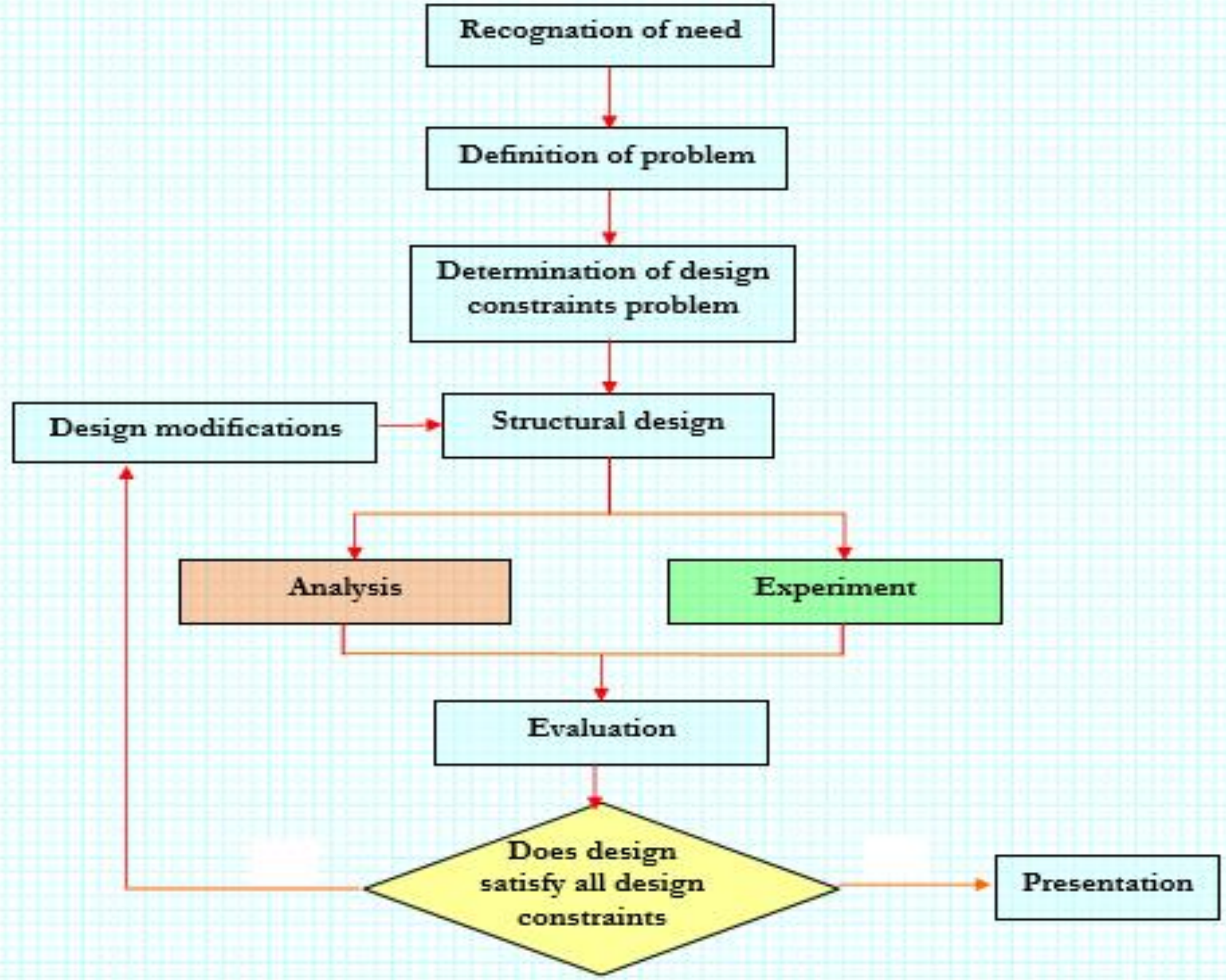
Common FEA Applications

- Mechanical/Aerospace/Civil/Automotive Engineering
- Structural/Stress Analysis
  - Static/Dynamic
  - Linear/Nonlinear
- Fluid Flow
- Heat Transfer
- Electromagnetic Fields
- Soil Mechanics
- Acoustics
- Biomechanics

Advantages

- Irregular Boundaries
- General Loads
- Different Materials
- Boundary Conditions
- Variable Element Size
- Easy Modification
- Dynamics
- Nonlinear Problems (Geometric or Material)

DESIGN PROCESS AND FEA



EXPERIMENT OR ANALYSIS?

EXPERIMENT

- Trial-error approach
- Expensive for the large systems
- Failure during the experiment may be dangerous

ANALYSIS

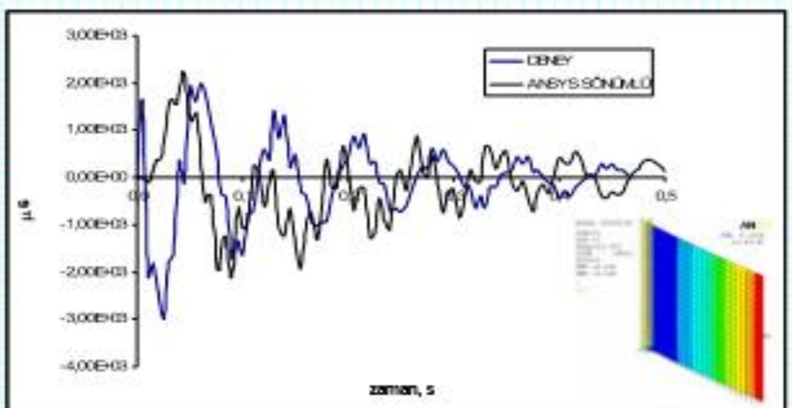
- No need for actual structure
- Assumptions and approximations affects adequacy of the results.

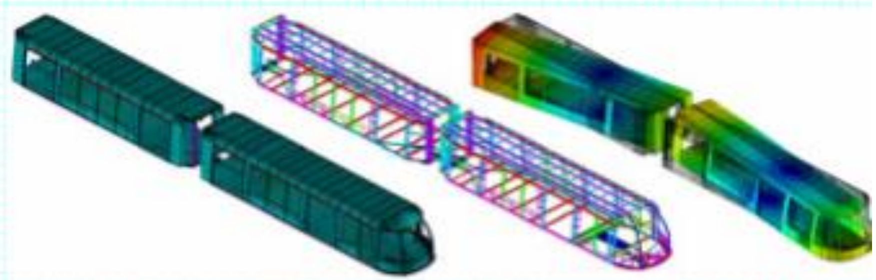
Analysis and experiment are dispensable in the design process.

EXPERIMENT AND ANALYSIS EXAMPLES

EXPERIMENTS

ANALYSES

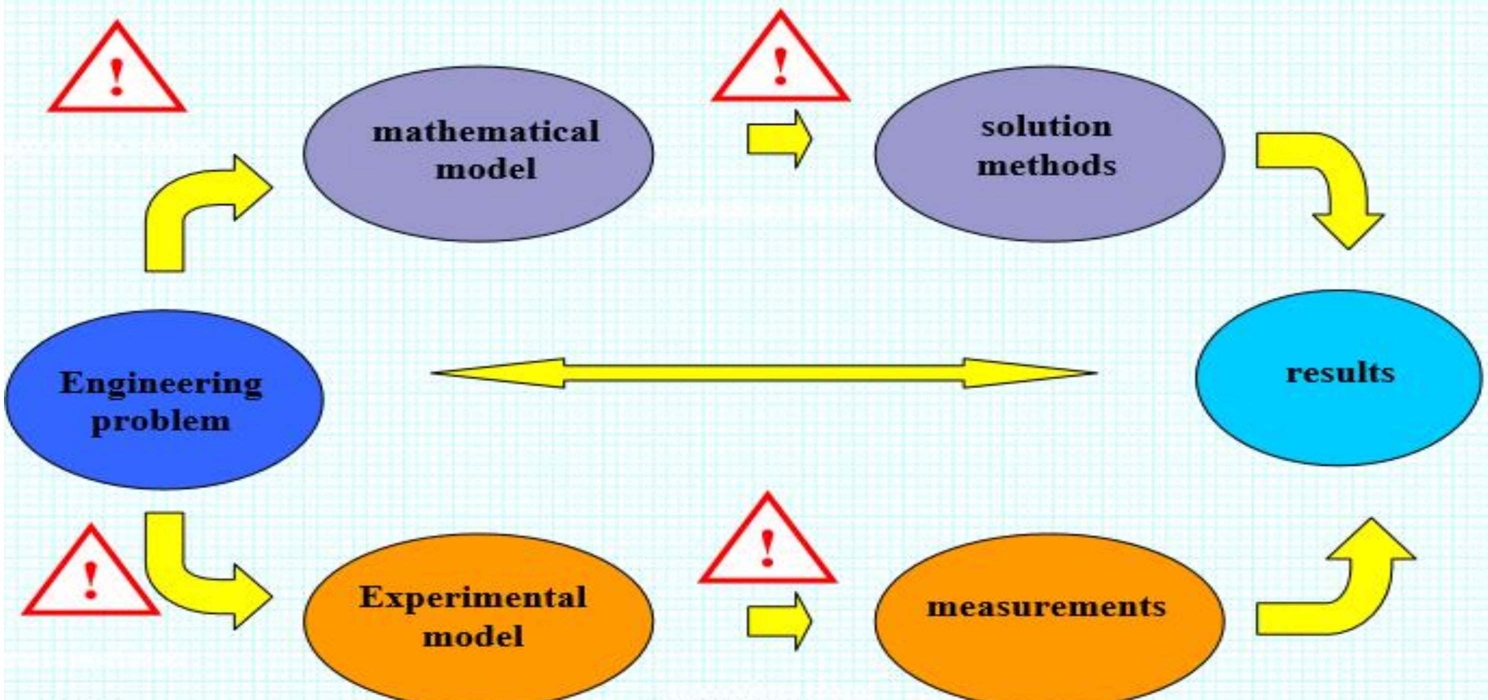




MODELING

- An analysis method is applied to a model problem rather than to an actual physical problem.
- Even laboratory experiments use models unless the actual physical structure is tested.

...MODELING



SOLUTION METHODS

Analytical Solution Methods

- They provide closed form exact solutions to the mathematical model of engineering problems.
- They can be used only if the geometry, loading and boundary conditions of the problem are simple.

Approximate Solution Methods

- They provide closed form approximate solutions to the mathematical model of engineering problems.
- They can be used only if the geometry, loading and boundary conditions of the problem are simple.

Numerical Solution Methods

- They provide discrete form approximate solution to the mathematical model of engineering problems.
- They can be used to solve the problems with relatively complex geometry, loading and boundary conditions.

  In particular finite elements can represent structures of arbitrarily complex geometry.

ADVANTAGES OF FEA

- Versatility: FEA is applicable to any field problem, such as heat transfer, stress analysis, magnetic fields and so on.
- There is no geometric restriction (The body or region may have any shape).
- Boundary conditions and loading are not restricted (boundary conditions and loads may be applied to any portion)

STEPS OF FEA

1. Learning about the problem
2. Modeling the problem
3. Preliminary Analysis
4. Discretizing the model
5. Formulating the solution
6. Obtaining Linear Algebraic Equations (LEAs)
7. Solving the equations
8. Checking the results

1. Learning about the problem

It is important to understand the physics or nature of the problem and classify it.

PROBLEM IDENTIFICATION

- What are the more important physical phenomena involved?
- Is the problem time-independent or time-dependent? (static or dynamic?)
- Is nonlinearity involved? (Is iterative solution necessary or not?)
- What results are sought from analysis?
- What accuracy is required?
- Is the problem interdisciplinary?

…Learning about the problem

From answers

i) necessary information to carry out an analysis

ii) how the problem is modeled

iii) what method of solution is adopted

- You must decide to do a nonlinear analysis if stresses are high enough to produce yielding.
- You must decide to perform a buckling analysis if the thin sections carry compressive load.

…Learning about the problem

Cautions:

- Without this step a proper model cannot be devised.
- At present, software does not automatically decide what solution procedure must apply to the problem.
- Software has limitations and almost contains errors.
- Yet the engineer, not to software provider, is legally responsible for results obtained.

2. Modeling the problem

- FEA is simulation, not reality.
- FEA is also applied to the mathematical model.
- If the model is inappropriate or inadequate, even very accurate FEA may show some disagreements with physical reality.

...Modeling the problem

STEPS

- Understand the physical nature of the problem.
- Exclude superfluous detail but include all essential features. (Unnecessary detail can be omitted)
- Make necessary assumptions for geometry, material, and loading. (i.e. fixed support, concentrated load, linear elastic material assumptions)
- Devise a model problem for the analysis.
- Write down the differential equations and boundary conditions, or integral statement.

...Modeling the problem

MODEL SELECTION

- What theory or mathematical formulation describes behavior?
- Depending on the dimensions, loading, and boundary conditions of this idealization

  we may decide that behavior is described by

  beam theory

  plate-bending theory

  equations of plane elasticity

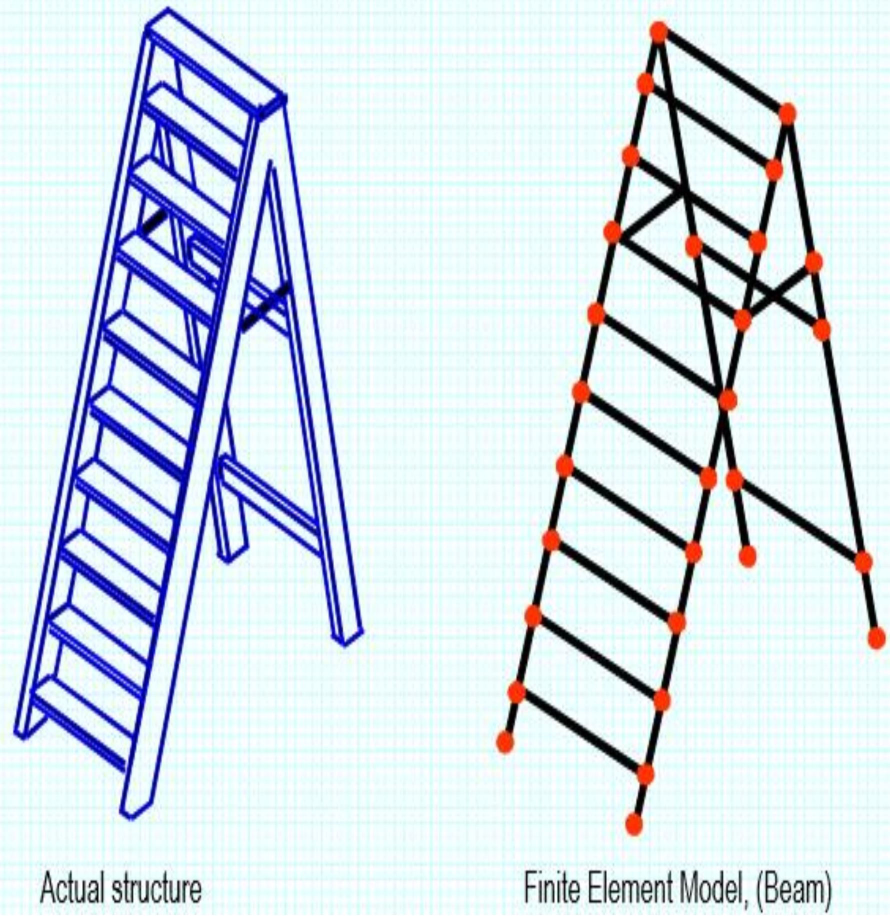
  or some other analysis theory.

...Modeling the problem

SELECTION FACTORS

Modeling decisions are influenced by

- what information is sought
- what accuracy is required
- the anticipated expense of FEA
- its capabilities and limitations.



3. Preliminary analysis

- Before going from a mathematical model to FEA, at least one preliminary solution should be obtained.
- We may use whatever means are conveniently available.
  - simple analytical calculations,
  - handbook formulas,
  - trusted previous solutions,
  - or experiment.
- Evaluation of the preliminary analysis results may require a better mathematical model.

4. Discretizating the model

- A mathematical model is discretized by dividing it into a mesh of finite elements.
- Thus a fully continuous field is represented by a piecewise continuous field.
- A continuum problem is one with an infinite number of unknowns.
- The FE discretization procedures reduce the problem to one of finite number of unknowns.

...Discretizating the model

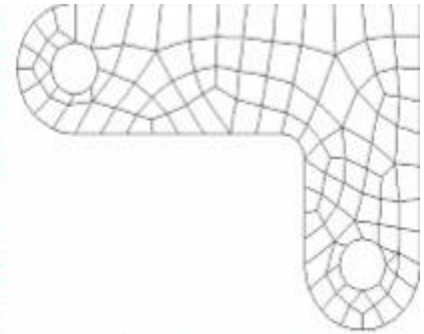
ERRORS

- Discretization introduces another approximation. Relative to reality, two sources of error have now been introduced:
  - § modeling error
  - § discretization error
- To reduce the errors

  improve model

  use more elements
- Numerical error is due to finite precision to represent data and the results manipulation.

...Discretizating the model

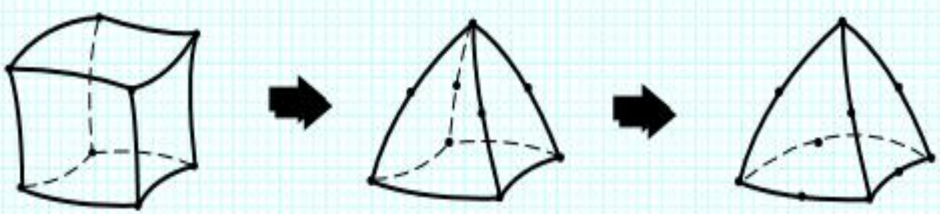


ELEMENTS AND NODES

By means of this method,

- Solution region is divided into a finite number of subregions (elements) of simple geometry (triangles, rectangles …)

- Key points are selected on the elements to serve as nodes.



8-Node Hex 9-Node Pyramid 10-Node Tet

...Discretizating the model

- The nodes usually lie on the element boundaries,
- Some elements have also a few interior nodes.
- The nodes share values of the field quantity
- They may also share its one or more derivatives.
- The nodal values are called as the degrees of freedom (d.o.f).
- The nodes are also locations where loads are applied and boundary conditions are imposed.

INTERPOLATION FUNCTIONS

- The FEA is an approximation based on piecewise interpolation of field quantity.
- The interpolation functions approximate (represent) the field variable in terms of the d.o.f. over a finite element.
- In this way, the problem is stated in terms of these nodal values as new unknowns.

- Polynomials are usually chosen as interpolation functions because differentiation and integration is easy with polynomials.

- The degree of polynomial depends on the number of unknowns at each node and certain compatibility and continuity requirements.

- Often functions are chosen so that the field variable and its derivatives are continuous across adjoining element boundaries.

5. Formulating the solution

- Now, we can formulate the solution for individual elements.
- There are four different approaches to formulate the properties of individual elements.

  Direct approach

  Variational approach

  Weighted residuals approach

  Energy balance approach
- Stiffness and equivalent nodal loads for a typical element are determined using the mentioned above.

6-7. Obtaining LEAs and solution

- The element properties are assembled to obtain the system equations.
- The equations are modified to account for the boundary conditions of the problem.
- The nodal displacements are obtained solving this simultaneous linear algebraic equation system.
- Support reactions are determined at restrained nodes.

- The strains and stresses are obtained from the element formulations using the nodal displacements.
- Once the nodal values (unknowns) are found, the interpolation functions define the field variable through the assemblage of elements.
- The nature of solution and the degree of approximation depend on the size and number of elements, and interpolation functions.

FINITE ELEMENT ANALYSIS

- PREPROCESSING
- NUMERICAL ANALYSIS
- POSTPROCESSING

NUMERICAL ANALYSIS

- Software automatically generates matrices that describe the behavior of each element,
- and combines these matrices into a large matrix equation that represents the FE structure,
- solves this equation to determine values of field quantities at nodes and additional calculations for nonlinear or time-dependent behavior.

PREPROCESSING

- Input data describes geometry, material properties, loads, and boundary conditions.
- Software can automatically prepare much of the FE mesh, but must be given direction as to the type of element and mesh density desired.
- Review the data for correctness before proceeding.

POSTPROCESSING

The FEA solution and quantities derived from it are listed or graphically displayed.

- Deformed shape (with exaggeration)
- Animation
- Stress of various types on various planes.

CHECK RESULTS

Cautions:

- Software has limitations and almost contains errors.
- Yet the engineer, not to software provider, is legally responsible for results obtained.

Examine results qualitatively

- Are the boundary conditions applied correctly?
- Are the symmetries seen on the results?

Compare FEA results with solutions from preliminary analysis and with any other useful information that may be available.

Expect to revise:

- Rarely is the first FE analysis satisfactory.
- Obvious blunders must be corrected.
- Either physical understanding or the FE model, or both, may be at fault.
- Disagreements must be satisfactorily resolved by repair of the mathematical model and/or the FE model.

Method of Solutions

A. Classical methods (Analytical)

They offer a high degree of insight, but the problems are difficult or impossible to solve for anything but simple geometries and loadings.

B. Numerical methods

(I) Energy: Minimize an expression for the potential energy of the structure over the whole domain.

(II) Boundary element: Approximates functions satisfying the governing differential equations not the boundary conditions.

(III) Finite difference: Replaces governing differential equations and boundary conditions with algebraic finite difference equations.

(IV) Finite element: Approximates the behavior of an irregular, continuous structure under general loadings and constraints with an assembly of discrete elements.

Common FEA Applications

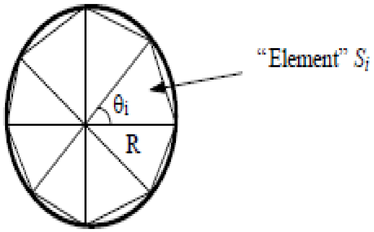
- Mechanical/Aerospace/Civil/Automotive Engineering
- Structural/Stress Analysis
  - Static/Dynamic
  - Linear/Nonlinear
- Fluid Flow
- Heat Transfer
- Electromagnetic Fields
- Soil Mechanics
- Acoustics
- Biomechanics

Advantages

- Irregular Boundaries
- General Loads
- Different Materials
- Boundary Conditions
- Variable Element Size
- Easy Modification
- Dynamics
- Nonlinear Problems (Geometric or Material)

➢The ***finite element method (FEM)***, *or* ***finite element analysis* (FEA)**, is based on the idea of building a complicated object with simple blocks, or, dividing a complicated object into small and manageable pieces.

➢Application of this simple idea can be found everywhere in everyday life as well as in engineering.



Area of one triangle: $S_i = \frac{1}{2} R^2 \sin\theta_i$

Area of the circle: $S_N = \sum_{i=1}^{N} S_i = \frac{1}{2} R^2 N \sin\left(\frac{2\pi}{N}\right) \rightarrow \pi R^2 \text{ as } N \rightarrow \infty$

where N = total number of triangles (elements).

Examples:

- Lego (kids' play)
- Buildings
- Approximation of the area of a circle:

Why Finite Element Method?

- *Design analysis: hand calculations, experiments, and* computer simulations
- FEM/FEA is the most widely applied computer simulation method in engineering
- Closely integrated with CAD/CAM applications
- ...

Applications of FEM in Engineering

- Mechanical/Aerospace/Civil/Automobile Engg.
- Structure analysis (static/dynamic, linear/nonlinear)
- Thermal/fluid flows
- Electromagnetics
- Geomechanics
- Biomechanics

... Applied sciences, BVP, Diff. Eq.prob (Laplace. Poison, heat cond.

II. Review of Matrix Algebra for Solution of FEM

II. Review of Matrix Algebra

Linear System of Algebraic Equations

$$\begin{aligned} a_{11}x_1 + a_{12}x_2 + \ldots + a_{1n}x_n &= b_1 \\ a_{21}x_1 + a_{22}x_2 + \ldots + a_{2n}x_n &= b_2 \\ \ldots\ldots\ldots & \\ a_{n1}x_1 + a_{n2}x_2 + \ldots + a_{nn}x_n &= b_n \end{aligned} \qquad (1)$$

where $x_1, x_2, \ldots, x_n$ are the unknowns.

In *matrix form*:

$$\mathbf{Ax} = \mathbf{b} \qquad (2)$$

where

$$\mathbf{A} = [a_{ij}] = \begin{bmatrix} a_{11} & a_{12} & \cdots & a_{1n} \\ a_{21} & a_{22} & \cdots & a_{2n} \\ \cdots & \cdots & \cdots & \cdots \\ a_{n1} & a_{n2} & \cdots & a_{nn} \end{bmatrix}$$

$$\mathbf{x} = \{x_i\} = \begin{Bmatrix} x_1 \\ x_2 \\ \vdots \\ x_n \end{Bmatrix} \qquad \mathbf{b} = \{b_i\} = \begin{Bmatrix} b_1 \\ b_2 \\ \vdots \\ b_n \end{Bmatrix} \qquad (3)$$

A is called a $n \times n$ (square) matrix, and **x** and **b** are (column) vectors of dimension n.

Row and Column Vectors

$$\mathbf{v} = [v_1 \quad v_2 \quad v_3] \qquad \mathbf{w} = \begin{Bmatrix} w_1 \\ w_2 \\ w_3 \end{Bmatrix}$$

Matrix Addition and Subtraction

For two matrices **A** and **B**, both of the *same size* ($m \times n$), the addition and subtraction are defined by

$$\mathbf{C} = \mathbf{A} + \mathbf{B} \qquad \text{with} \quad c_{ij} = a_{ij} + b_{ij}$$

$$\mathbf{D} = \mathbf{A} - \mathbf{B} \qquad \text{with} \quad d_{ij} = a_{ij} - b_{ij}$$

Scalar Multiplication

$$\lambda\mathbf{A} = [\lambda a_{ij}]$$

Matrix Multiplication

For two matrices **A** (of size $l \times m$) and **B** (of size $m \times n$), the product of **AB** is defined by

$$\mathbf{C} = \mathbf{AB} \qquad \text{with } c_{ij} = \sum_{k=1}^{m} a_{ik} b_{kj}$$

where $i = 1, 2, \ldots, l$; $j = 1, 2, \ldots, n$.

Note that, in general, $\mathbf{AB} \neq \mathbf{BA}$, but $(\mathbf{AB})\mathbf{C} = \mathbf{A}(\mathbf{BC})$ (associative).

Transpose of a Matrix

If $\mathbf{A} = [a_{ij}]$, then the transpose of $\mathbf{A}$ is

$$\mathbf{A}^T = \left[a_{ji}\right]$$

Notice that $(\mathbf{AB})^T = \mathbf{B}^T\mathbf{A}^T$.

Symmetric Matrix

A *square* $(n \times n)$ matrix $\mathbf{A}$ is called symmetric, if

$$\mathbf{A} = \mathbf{A}^T \qquad \text{or} \qquad a_{ij} = a_{ji}$$

Unit (Identity) Matrix

$$\mathbf{I} = \begin{bmatrix} 1 & 0 & \dots & 0 \\ 0 & 1 & \dots & 0 \\ \dots & \dots & \dots & \dots \\ 0 & 0 & \dots & 1 \end{bmatrix}$$

Note that $\mathbf{AI} = \mathbf{A}$, $\mathbf{Ix} = \mathbf{x}$.

Determinant of a Matrix

The determinant of *square* matrix $\mathbf{A}$ is a scalar number denoted by $\det \mathbf{A}$ or $|\mathbf{A}|$. For 2×2 and 3×3 matrices, their determinants are given by

$$\det\begin{bmatrix} a & b \\ c & d \end{bmatrix} = ad - bc$$

$$\det\begin{bmatrix} a_{11} & a_{12} & a_{13} \\ a_{21} & a_{22} & a_{23} \\ a_{31} & a_{32} & a_{33} \end{bmatrix} = a_{11}a_{22}a_{33} + a_{12}a_{23}a_{31} + a_{21}a_{32}a_{13}$$
$$- a_{13}a_{22}a_{31} - a_{12}a_{21}a_{33} - a_{23}a_{32}a_{11}$$

Singular Matrix

A *square* matrix $\mathbf{A}$ is *singular* if $\det \mathbf{A} = 0$, which indicates problems in the systems (nonunique solutions, degeneracy, etc.)

Matrix Inversion

For a *square* and *nonsingular* matrix $\mathbf{A}$ ($\det \mathbf{A} \neq 0$), its *inverse* $\mathbf{A}^{-1}$ is constructed in such a way that

$$\mathbf{AA}^{-1} = \mathbf{A}^{-1}\mathbf{A} = \mathbf{I}$$

The *cofactor matrix* $\mathbf{C}$ of matrix $\mathbf{A}$ is defined by

$$C_{ij} = (-1)^{i+j} M_{ij}$$

where M_{ij} is the determinant of the smaller matrix obtained by eliminating the ith row and jth column of **A**.

Thus, the inverse of **A** can be determined by

$$\mathbf{A}^{-1} = \frac{1}{\det \mathbf{A}} \mathbf{C}^T$$

We can show that $(\mathbf{AB})^{-1} = \mathbf{B}^{-1}\mathbf{A}^{-1}$.

Examples:

(1) $$\begin{bmatrix} a & b \\ c & d \end{bmatrix}^{-1} = \frac{1}{(ad-bc)} \begin{bmatrix} d & -b \\ -c & a \end{bmatrix}$$

Checking,

$$\begin{bmatrix} a & b \\ c & d \end{bmatrix}^{-1} \begin{bmatrix} a & b \\ c & d \end{bmatrix} = \frac{1}{(ad-bc)} \begin{bmatrix} d & -b \\ -c & a \end{bmatrix} \begin{bmatrix} a & b \\ c & d \end{bmatrix} = \begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix}$$

(2) $$\begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 2 \end{bmatrix}^{-1} = \frac{1}{(4-2-1)} \begin{bmatrix} 3 & 2 & 1 \\ 2 & 2 & 1 \\ 1 & 1 & 1 \end{bmatrix}^T = \begin{bmatrix} 3 & 2 & 1 \\ 2 & 2 & 1 \\ 1 & 1 & 1 \end{bmatrix}$$

Checking,

$$\begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 2 \end{bmatrix} \begin{bmatrix} 3 & 2 & 1 \\ 2 & 2 & 1 \\ 1 & 1 & 1 \end{bmatrix} = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix}$$

If $\det \mathbf{A} = 0$ (i.e., **A** is singular), then $\mathbf{A}^{-1}$ does not exist!

The solution of the linear system of equations (Eq.(1)) can be expressed as (assuming the coefficient matrix **A** is nonsingular)

$$\mathbf{x} = \mathbf{A}^{-1}\mathbf{b}$$

Thus, the main task in solving a linear system of equations is to found the inverse of the coefficient matrix.

Solution Techniques for Linear Systems of Equations

- Gauss elimination methods
- Iterative methods

Positive Definite Matrix

A *square* $(n \times n)$ matrix $\mathbf{A}$ is said to be *positive definite*, if for any nonzero vector $\mathbf{x}$ of dimension n,

$$\mathbf{x}^T \mathbf{A} \mathbf{x} > 0$$

Note that positive definite matrices are nonsingular.

Differentiation and Integration of a Matrix

Let

$$\mathbf{A}(t) = \left[a_{ij}(t) \right]$$

then the differentiation is defined by

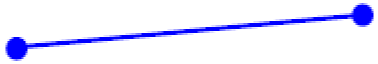
$$\frac{d}{dt}\mathbf{A}(t) = \left[\frac{da_{ij}(t)}{dt} \right]$$

and the integration by

$$\int \mathbf{A}(t)dt = \left[\int a_{ij}(t)dt \right]$$

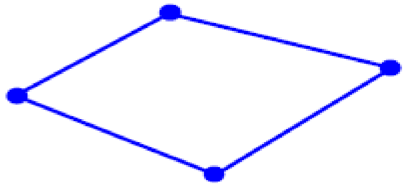
Types of Finite Elements

1-D (Line) Element



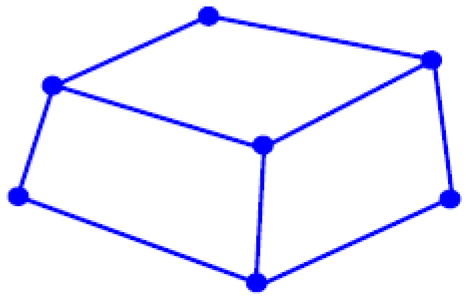
(Spring, truss, beam, pipe, etc.)

2-D (Plane) Element



(Membrane, plate, shell, etc.)

3-D (Solid) Element



(3-D fields - temperature, displacement, stress, flow velocity)

Procedure for solution of FEM problem

Step 1 Select the Element Type

Step 2 Select a Displacement Function

Step 3 Define the Strain/Displacement and Stress/Strain Relationships

Step 4 Derive the Element Stiffness Matrix and Equations

Step 5 Assemble the Element Equations to Obtain the Global Equations and Introduce Boundary Conditions

Step 6 Solve for the Nodal Displacements

Step 7 Solve for the Element Forces

Convergence – Cont'd

❖ Types of Refinement:

• ***h-refinement*****:** reduce the size of the element ("*h"* refers to the typical size of the elements);

• ***p-refinement*****:** Increase the order of the polynomials on an element (linear to quadratic, etc.; "*h*" refers to the highest order in a polynomial);

• ***r-refinement*****:** re-arrange the nodes in the mesh;

• ***hp-refinement*****:** Combination of the h- and p-refinements (better results!).

❖ Types of Errors:

· **Modeling Error** (beam, plate … theories)

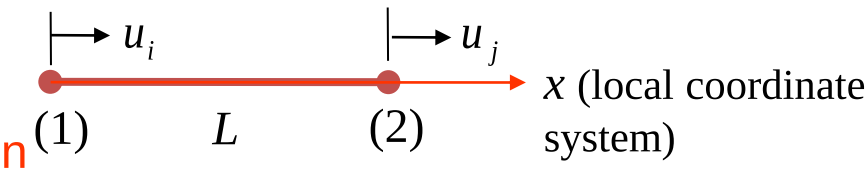
· **Discretization Error** (finite, piecewise …)

· **Numerical Error** (in solving FE equations)

Isoparmetric formulation of bar element stiffness matrix

Recall 1-D bar element with local coordinate system

Step 1 Select the Element Type



Step 2 Select a Displacement Function

Approximate Elastic Displacement

$$u = a_1 + a_2 x \quad \Rightarrow \quad \begin{array}{l} u_1 = a_1 \\ u_2 = a_1 + a_2 L \end{array}$$

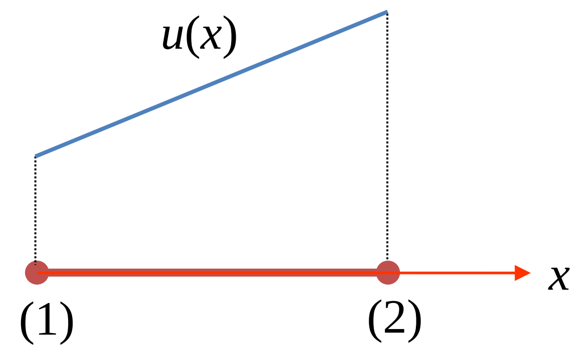
$$\Rightarrow u = u_1 + \frac{u_2 - u_1}{L} x = \left(1 - \frac{x}{L}\right) u_1 + \left(\frac{x}{L}\right) u_2$$

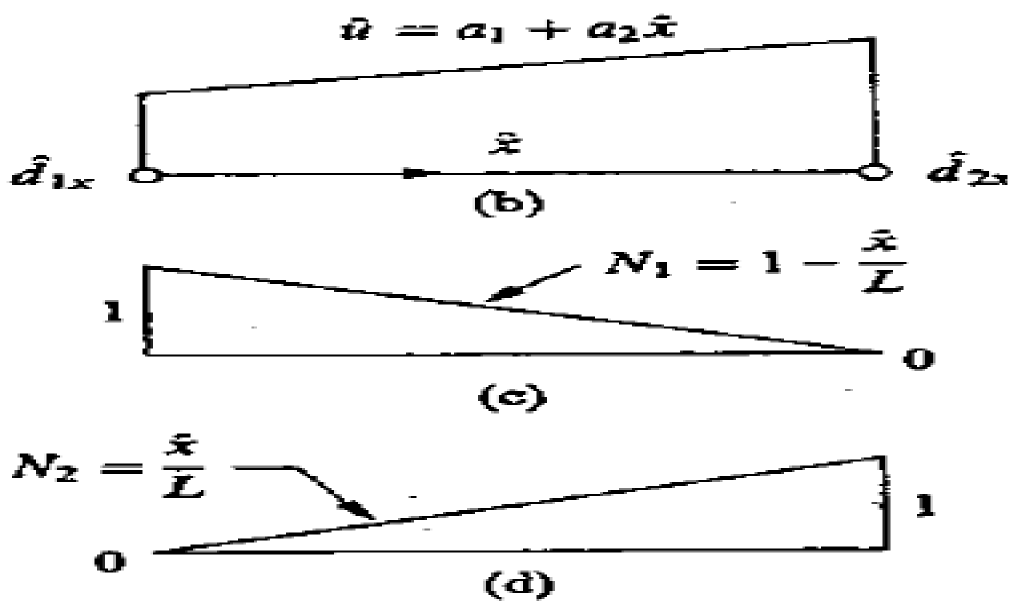
$$= \psi_1(x) u_1 + \psi_2(x) u_2$$

$$u = \begin{bmatrix} \psi_1 & \psi_2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{bmatrix} 1 - \frac{x}{L} & \frac{x}{L} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = [\mathbf{N}]\{\mathbf{d}\}$$

$[\mathbf{N}]$ = Approximation Function Matrix

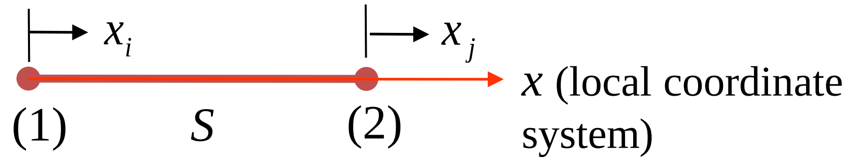
$\{\mathbf{d}\}$ = Nodal Displacement Vector





Isoparmetric formulation of bar element stiffness matrix

1-D bar element with natural coordinate system



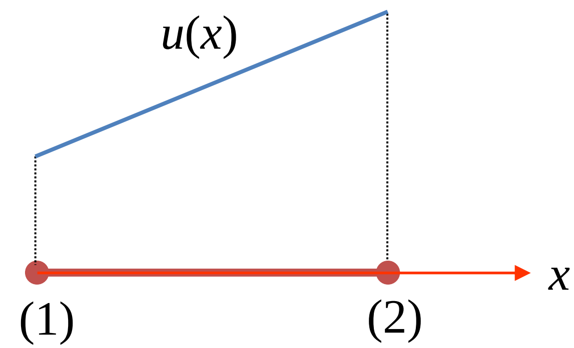
Approximate Elastic Displacement

$$x = a_1 + a_2 s \quad \Rightarrow \quad with\ s\ varies\ from -1\ to +1$$

$$\Rightarrow\ x = 1/2[(1-s)x_1 + (1+s)x_2]$$

$$= N_1(x)x_1 + N_2(x)x_2$$

$$x = \begin{bmatrix} N_1 & N_2 \end{bmatrix} \begin{Bmatrix} x_1 \\ x_2 \end{Bmatrix} = \begin{bmatrix} \dfrac{1-s}{2} & \dfrac{1+s}{2} \end{bmatrix} \begin{Bmatrix} x_1 \\ x_2 \end{Bmatrix} = [\mathbf{N}]\{\mathbf{d}\}$$



$[\mathbf{N}]$ = Approximation Function Matrix

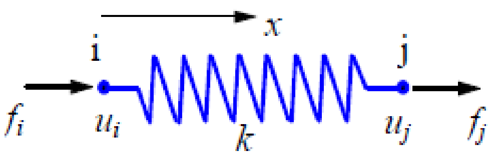
$\{\mathbf{d}\}$ = Nodal Displacement Vector

Figure 10–1 Linear bar element in (a) a global coordinate system x and (b) a natural coordinate system s

III. Spring Element

"Everything important is simple."

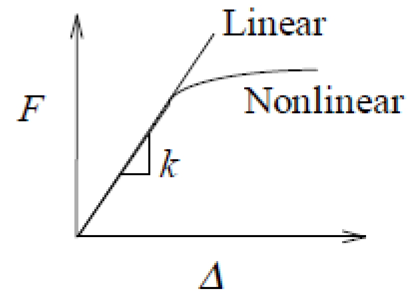
One Spring Element



Two nodes:	i, j
Nodal displacements:	u_i, u_j (in, m, mm)
Nodal forces:	f_i, f_j (lb, Newton)
Spring constant (stiffness):	k (lb/in, N/m, N/mm)

Spring force-displacement relationship:

$$F = k\Delta \qquad \text{with } \Delta = u_j - u_i$$



$k = F / \Delta$ (> 0) is the force needed to produce a unit stretch.

Governing Equation

$$\mathbf{ku} = \mathbf{f}$$

where

$\mathbf{k}$ = (element) stiffness matrix

$\mathbf{u}$ = (element nodal) displacement vector

$\mathbf{f}$ = (element nodal) force vector

Note that $\mathbf{k}$ is symmetric. Is $\mathbf{k}$ singular or nonsingular? That is, can we solve the equation? If not, why?

Consider the equilibrium of forces for the spring. At node i, we have

$$f_i = -F = -k(u_j - u_i) = ku_i - ku_j$$

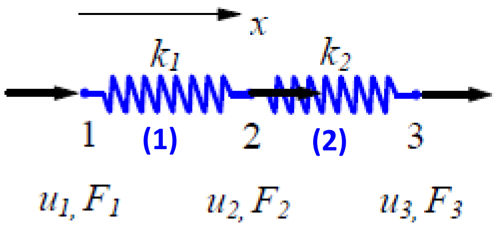
d at node j,

$$f_j = F = k(u_j - u_i) = -ku_i + ku_j$$

matrix form,

$$\begin{bmatrix} k & -k \\ -k & k \end{bmatrix} \begin{Bmatrix} u_i \\ u_j \end{Bmatrix} = \begin{Bmatrix} f_i \\ f_j \end{Bmatrix}$$

Two spring system



For element 1, **i.e. (1)**

$$\begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} f_1^1 \\ f_2^1 \end{Bmatrix}$$

element 2, **i.e. (2)**

$$\begin{bmatrix} k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} f_1^2 \\ f_2^2 \end{Bmatrix}$$

where f_i^m is the (internal) force acting on *local* node i of element m $(i = 1, 2)$.

Assemble the stiffness matrix for the whole system:

Consider the equilibrium of forces at node 1,

$$F_1 = f_1^1$$

at node 2,

$$F_2 = f_2^1 + f_1^2$$

and node 3,

$$F_3 = f_2^2$$

That is,

$$F_1 = k_1 u_1 - k_1 u_2$$
$$F_2 = -k_1 u_1 + (k_1 + k_2) u_2 - k_2 u_3$$
$$F_3 = -k_2 u_2 + k_2 u_3$$

In matrix form,

$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 + k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix}$$

Gov. Eq.

$$\mathbf{KU} = \mathbf{F}$$

K is the stiffness matrix (structure matrix) for the spring system.

An alternative way of assembling the whole stiffness matrix:

"Enlarging" the stiffness matrices for elements 1 and 2, we have

$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 & 0 \\ 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} f_1^1 \\ f_2^1 \\ 0 \end{Bmatrix}$$

$$\begin{bmatrix} 0 & 0 & 0 \\ 0 & k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ f_1^2 \\ f_2^2 \end{Bmatrix}$$

Adding the two matrix equations (*superposition*), we have

$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1+k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} f_1^1 \\ f_2^1 + f_1^2 \\ f_2^2 \end{Bmatrix}$$

This is the same equation we derived by using the force equilibrium concept.

Boundary and load conditions:

Assuming, $u_1 = 0$ and $F_2 = F_3 = P$

we have

$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1+k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} 0 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ P \\ P \end{Bmatrix}$$

which reduces to

$$\begin{bmatrix} k_1+k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} P \\ P \end{Bmatrix}$$

and

$$F_1 = -k_1 u_2$$

Unknowns are

$$\mathbf{U} = \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix}$$ and the reaction force F_1 (if desired).

Solving the equations, we obtain the displacements

$$\begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} 2P/k_1 \\ 2P/k_1 + P/k_2 \end{Bmatrix}$$

and the reaction force

$$F_1 = -2P$$

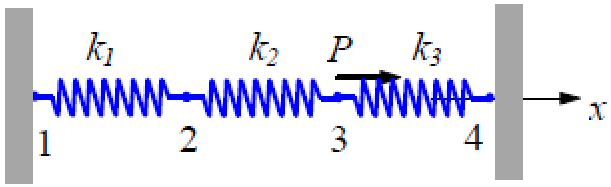
Checking the Results

- Deformed shape of the structure
- Balance of the external forces
- Order of magnitudes of the numbers

Notes About the Spring Elements

- Suitable for stiffness analysis
- Not suitable for stress analysis of the spring itself
- Can have spring elements with stiffness in the lateral direction, spring elements for torsion, etc.

Example 1.1



Given: For the spring system shown above,

$k_1 = 100$ N/mm, $k_2 = 200$ N/mm, $k_3 = 100$ N/mm

$P = 500$ N, $u_1 = u_4 = 0$

Find: (a) the global stiffness matrix

(b) displacements of nodes 2 and 3

(c) the reaction forces at nodes 1 and 4

(d) the force in the spring 2

Solution

(a) The element stiffness matrices are

$$\mathbf{k}_1 = \begin{bmatrix} 100 & -100 \\ -100 & 100 \end{bmatrix} \quad \text{(N/mm)} \qquad (1)$$

$$\mathbf{k}_2 = \begin{bmatrix} 200 & -200 \\ -200 & 200 \end{bmatrix} \quad \text{(N/mm)} \qquad (2)$$

$$\mathbf{k}_3 = \begin{bmatrix} 100 & -100 \\ -100 & 100 \end{bmatrix} \quad \text{(N/mm)} \qquad (3)$$

Applying the superposition concept, we obtain the global stiffness matrix for the spring system as

$$\mathbf{K} = \begin{array}{c} \begin{array}{cccc} u_1 & u_2 & u_3 & u_4 \end{array} \\ \begin{bmatrix} 100 & -100 & 0 & 0 \\ -100 & 100+200 & -200 & 0 \\ 0 & -200 & 200+100 & -100 \\ 0 & 0 & -100 & 100 \end{bmatrix} \end{array}$$

or

$$\mathbf{K} = \begin{bmatrix} 100 & -100 & 0 & 0 \\ -100 & 300 & -200 & 0 \\ 0 & -200 & 300 & -100 \\ 0 & 0 & -100 & 100 \end{bmatrix}$$

which is *symmetric* and ***banded***.

Equilibrium (FE) equation for the whole system is

$$\begin{bmatrix} 100 & -100 & 0 & 0 \\ -100 & 300 & -200 & 0 \\ 0 & -200 & 300 & -100 \\ 0 & 0 & -100 & 100 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ 0 \\ P \\ F_4 \end{Bmatrix} \qquad (4)$$

(b) Applying the BC ($u_1 = u_4 = 0$) in Eq(4), or deleting the 1st and 4th rows and columns, we have

$$\begin{bmatrix} 300 & -200 \\ -200 & 300 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ P \end{Bmatrix} \qquad (5)$$

Solving Eq.(5), we obtain

$$\begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} P/250 \\ 3P/500 \end{Bmatrix} = \begin{Bmatrix} 2 \\ 3 \end{Bmatrix} \text{ (mm)} \qquad (6)$$

(c) From the 1st and 4th equations in (4), we get the reaction forces

$$F_1 = -100u_2 = -200 \quad \text{(N)}$$

$$F_4 = -100u_3 = -300 \quad \text{(N)}$$

(d) The FE equation for spring (element) 2 is

$$\begin{bmatrix} 200 & -200 \\ -200 & 200 \end{bmatrix} \begin{Bmatrix} u_i \\ u_j \end{Bmatrix} = \begin{Bmatrix} f_i \\ f_j \end{Bmatrix}$$

Here $i = 2$, $j = 3$ for element 2. Thus we can calculate the spring force as

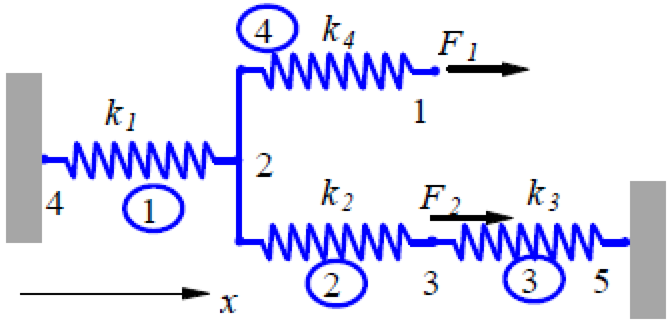
$$F = f_j = -f_i = \begin{bmatrix} -200 & 200 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix}$$

$$= \begin{bmatrix} -200 & 200 \end{bmatrix} \begin{Bmatrix} 2 \\ 3 \end{Bmatrix}$$

$$= 200 \quad \text{(N)}$$

Check the results ???

Example 1.2



Solution:

-Construct the following

Element Connectivity Table

Element	*Node i (1)*	*Node j (2)*
1	4	2
2	2	3
3	3	5
4	2	1

Which specify the global node numbers corresponding to the local node number

Element stiffness matrices

$$\mathbf{k}_1 = \begin{array}{cc} u_4 & u_2 \\ \left[\begin{matrix} k_1 & -k_1 \\ -k_1 & k_1 \end{matrix}\right. & \left.\right] \end{array}$$

$$\mathbf{k}_1 = \begin{array}{c} \begin{matrix} u_4 & u_2 \end{matrix} \\ \begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 \end{bmatrix} \end{array} \qquad \mathbf{k}_2 = \begin{array}{c} \begin{matrix} u_2 & u_3 \end{matrix} \\ \begin{bmatrix} k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix} \end{array}$$

$$\mathbf{k}_3 = \begin{array}{c} \begin{matrix} u_3 & u_5 \end{matrix} \\ \begin{bmatrix} k_3 & -k_3 \\ -k_3 & k_3 \end{bmatrix} \end{array} \qquad \mathbf{k}_4 = \begin{array}{c} \begin{matrix} u_2 & u_1 \end{matrix} \\ \begin{bmatrix} k_4 & -k_4 \\ -k_4 & k_4 \end{bmatrix} \end{array}$$

Apply superposition method obtain global K

$$\mathbf{K} = \begin{array}{c} \begin{matrix} u_1 & u_2 & u_3 & u_4 & u_5 \end{matrix} \\ \begin{bmatrix} k_4 & -k_4 & 0 & 0 & 0 \\ -k_4 & k_1+k_2+k_4 & -k_2 & -k_1 & 0 \\ 0 & -k_2 & k_2+k_3 & 0 & -k_3 \\ 0 & -k_1 & 0 & k_1 & 0 \\ 0 & 0 & -k_3 & 0 & k_3 \end{bmatrix} \end{array}$$

The matrix is symmetric, banded, but singular

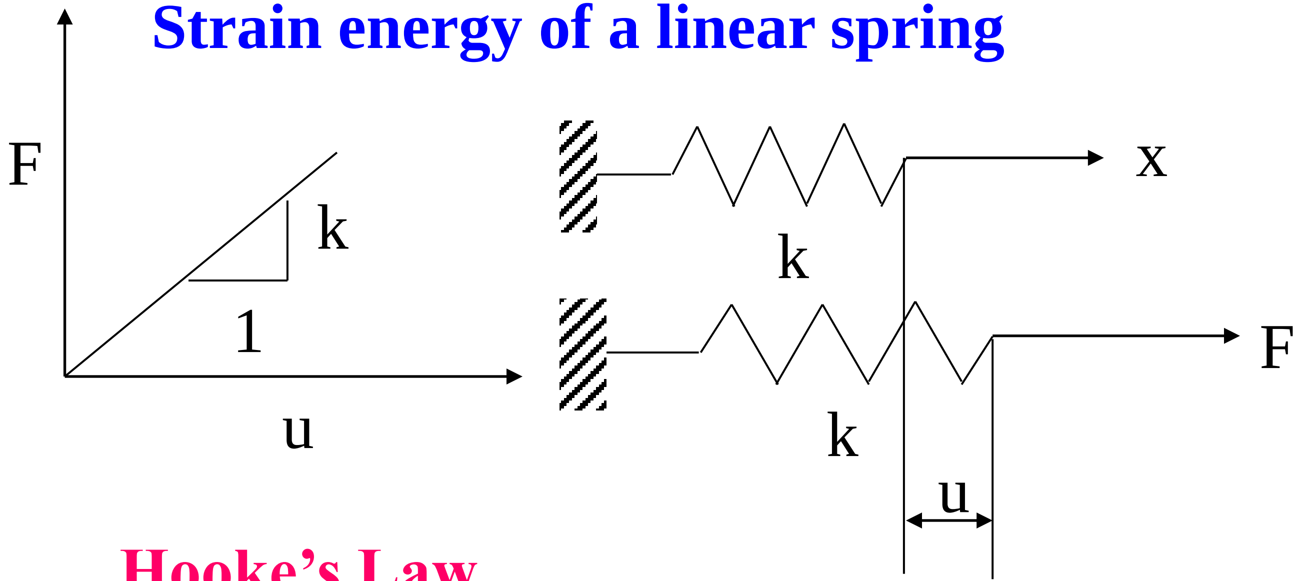
Potential energy Approach

At stable equilibrium, the body is identified by a minimum value of the total potential energy.

The **potential energy** of an elastic body is defined as

$$\boxed{\Pi = \text{Strain energy (U)} - \text{potential energy of loading (W)}}$$

Strain energy of a linear spring



F = Force in the spring

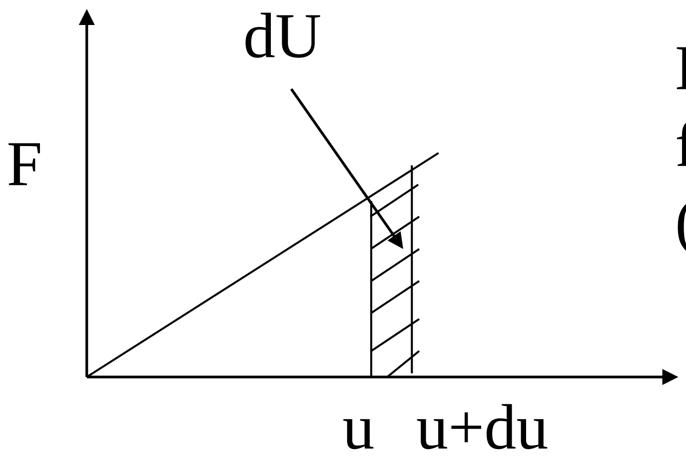
u = deflection of the spring

k = “stiffness” of the spring

Hooke’s Law

F = ku

Strain energy of a linear spring



Differential strain energy of the spring for a small change in displacement (du) of the spring

$$dU = Fdu$$

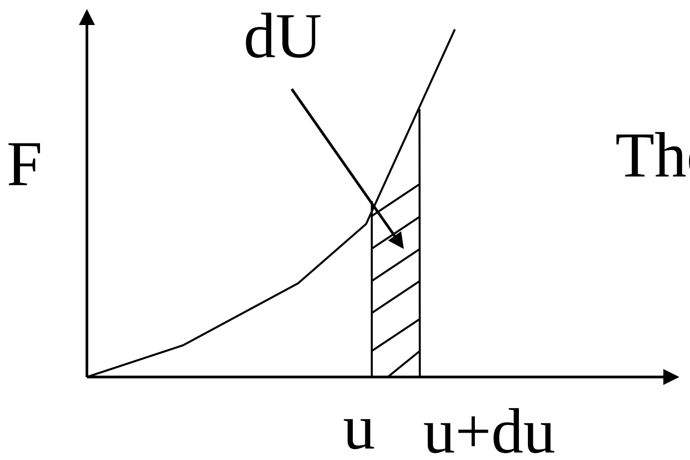
For a linear spring

$$dU = kudu$$

The total strain energy of the spring

$$U = \int_0^u k\, u\, du = \frac{1}{2} k\, u^2$$

Strain energy of a nonlinear spring



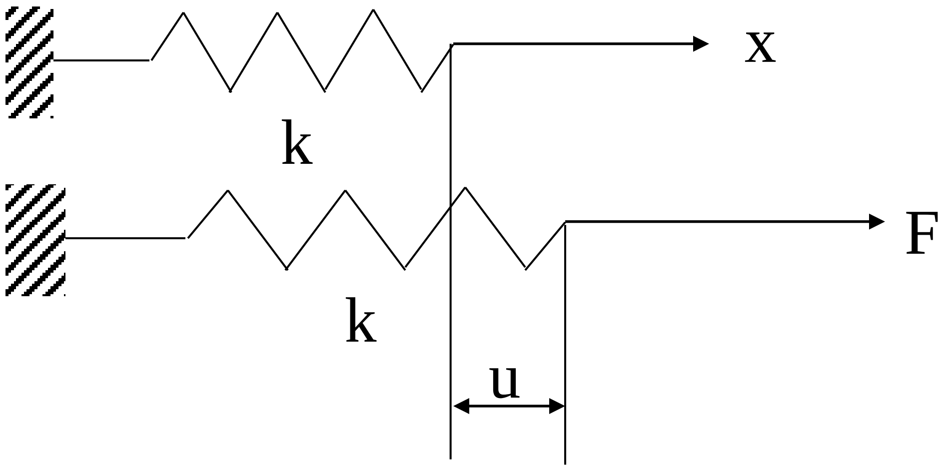
$$dU = F du$$

The total strain energy of the spring

$$U = \int_0^u F\,du = \text{Area under the force} - \text{dispalcement curve}$$

Potential energy of the loading (for a single spring as in the figure)

$$W = Fu$$



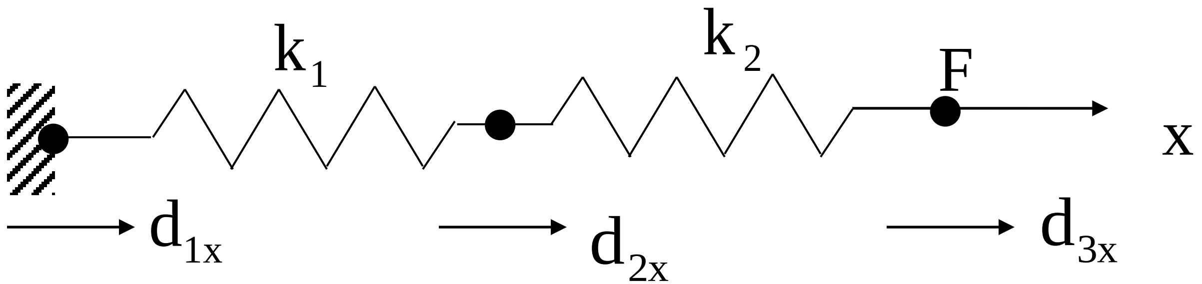
Potential energy of a linear spring

$$\boxed{\Pi = \text{Strain energy (U)} - \text{potential energy of loading}\ (W)}$$

$$\boxed{\Pi = \frac{1}{2}ku^2 - Fu}$$

Example of how to obtain the equlibr

Principle of <u>minimum potential energy</u> for a system of springs



For this system of spring, first write down the total potential energy of the system as:

$$\Pi_{system} = \left[\frac{1}{2}\mathrm{k}_1(\mathrm{d}_{2x})^2 + \frac{1}{2}\mathrm{k}_2(\mathrm{d}_{3x} - \mathrm{d}_{2x})^2\right] - \mathrm{Fd}_{3x}$$

Obtain the equilibrium equations by minimizing the potential energy

$$\frac{\partial \Pi_{system}}{\partial \mathrm{d}_{2x}} = \mathrm{k}_1\mathrm{d}_{2x} - \mathrm{k}_2(\mathrm{d}_{3x} - \mathrm{d}_{2x}) = 0 \qquad Equation\,(1)$$

$$\frac{\partial \Pi_{system}}{\partial \mathrm{d}_{3x}} = \mathrm{k}_2(\mathrm{d}_{3x} - \mathrm{d}_{2x}) - F = 0 \qquad Equation\,(2)$$

Principle of <u>minimum potential energy</u> for a system of springs

In matrix form, equations 1 and 2 look like

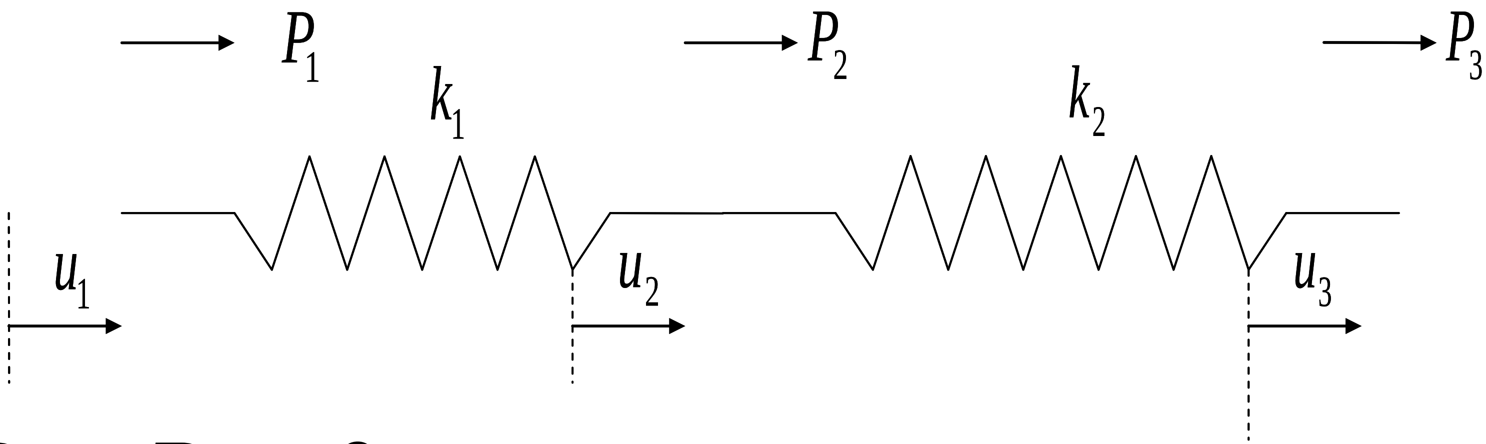
$$\begin{bmatrix} k_1 + k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix} \begin{bmatrix} d_{2x} \\ d_{3x} \end{bmatrix} = \begin{bmatrix} 0 \\ F \end{bmatrix}$$

Does this equation look familiar?

Also look at example problem <u>**worked out in class**</u>

STIFNESS AND FLEXIBILLITY. STIFFNES MATRIX

- The system is in equilibrium



$$P_1 + P_2 + P_3 = 0$$

$$P_1 = k_1(u_1 - u_2)$$
$$P_2 = k_2(u_3 - u_2)$$
$$P_2 = -k_1u_1 + (k_1 + k_2)u_2 - k_2u_3.$$

STIFNESS AND FLEXIBILLITY. STIFFNES MATRIX

- The equations written in matrix form:

$$\begin{Bmatrix} P_1 \\ P_2 \\ P_3 \end{Bmatrix} = \begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & (k_1 + k_2) & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix}$$

$$\{p\} = [K]\{u_i\}$$

- p- vector of external nodal loads acting on the structure
- K- system or structural stiffness matrix
- u-over-all nodal displacement vector

Thank You