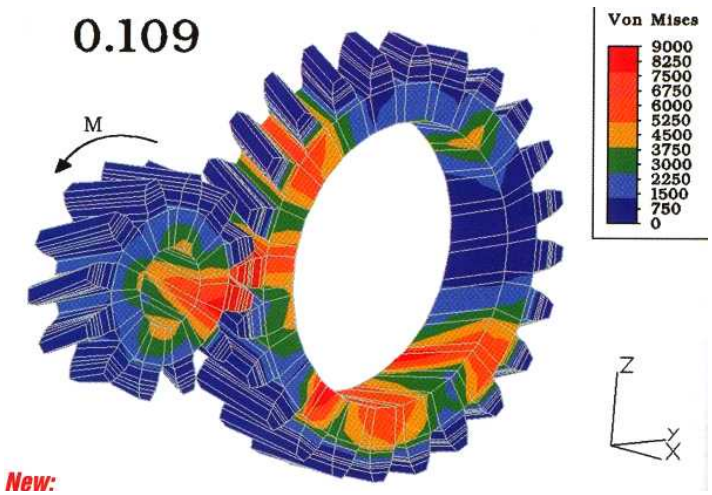
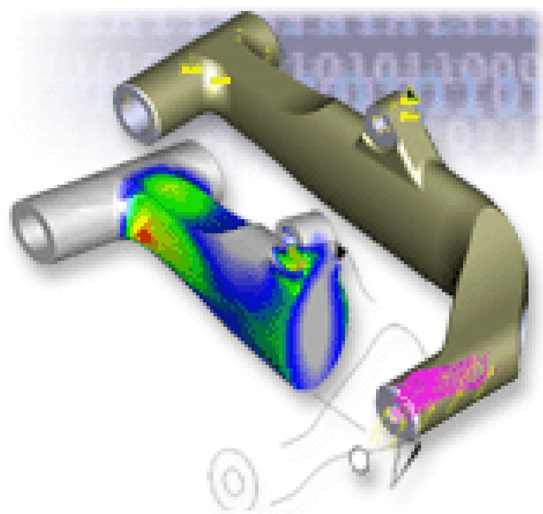
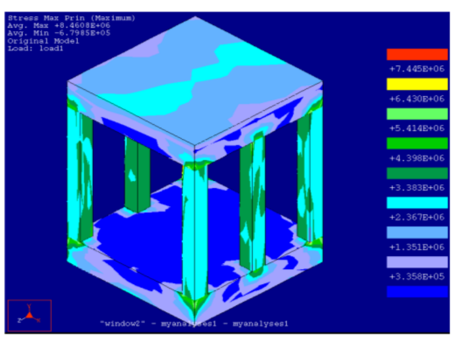
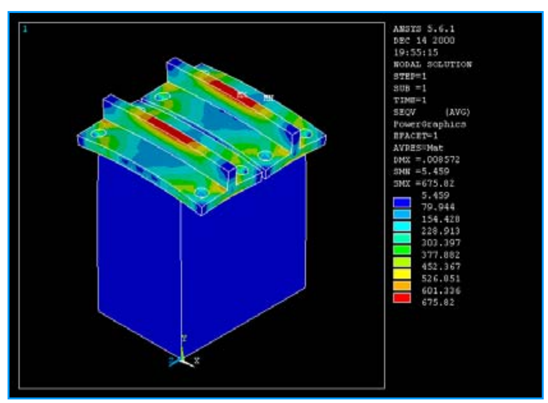
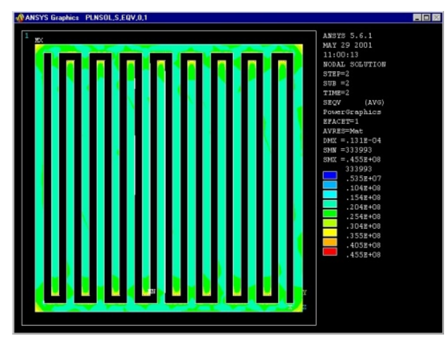
Introduction to Finite Element Analysis (FEA) or Finite Element Method (FEM)











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.**

The Purpose of FEA

Analytical Solution

- Stress analysis for trusses, beams, and other simple structures are carried out based on <u>dramatic simplification and idealization:</u>
  - 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 <u>idealized</u> structure & a large <u>*safety factor* (1.5-3)</u> 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 <u>complex</u> 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 <u>accurately</u>; and
  - To identify the optimal design with <u>confidence</u>

Brief History

- Grew out of aerospace industry
- Post-WW II jets, missiles, space flight
- Need for light weight structures
- Required accurate stress analysis
- Paralleled growth of computers

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

FEA

Discretization

Complex Object **Simple Analysis**

(Material discontinuity,

Complex and arbitrary geometry)

Real Word → Simplified (Idealized) Physical Model → Mathematical Model → Discretized (mesh) Model

Discretizations

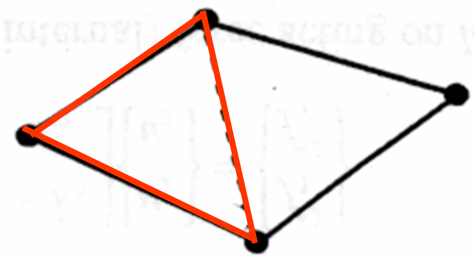
- Model body by dividing it into an equivalent system of many smaller bodies or units (finite elements) interconnected at points common to two or more elements (nodes or nodal points) and/or boundary lines and/or surfaces.

Types of Finite Elements

1-D (Line) Element

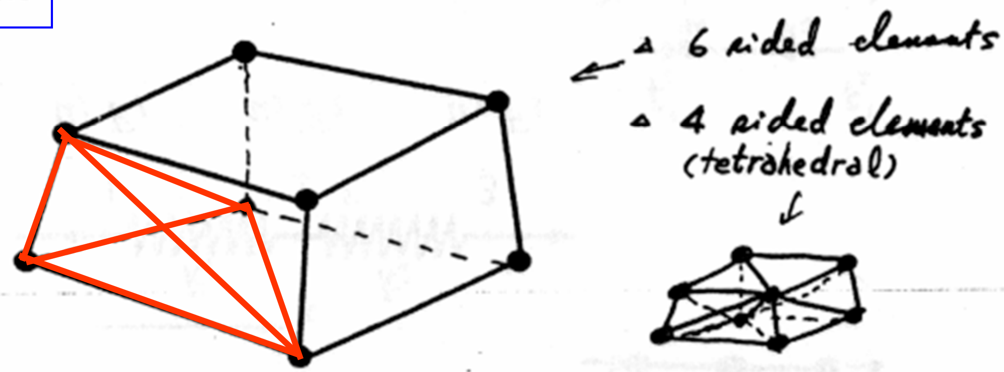
(Spring, truss, beam, pipe, etc.)

2-D (Plane) Element



(Membrane, plate, shell, etc.)

3-D (Solid) Element



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

Elements & Nodes - Nodal Quantity

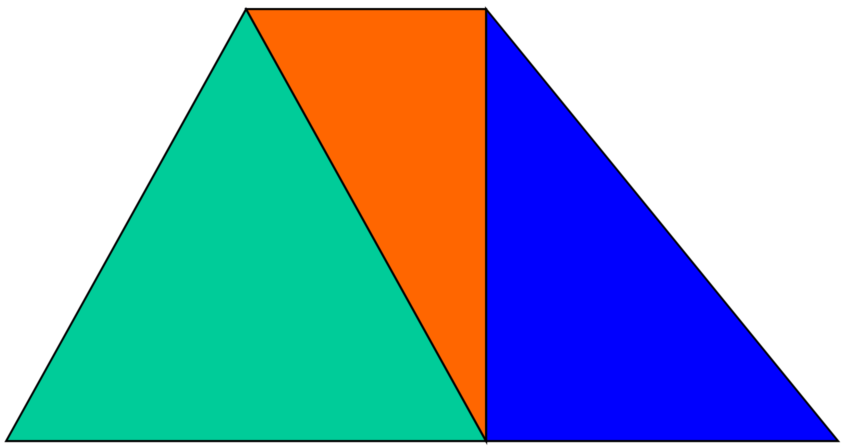
Feature

- **Obtain a set of algebraic equations to solve for unknown (first) <u>nodal quantity</u> (displacement).**

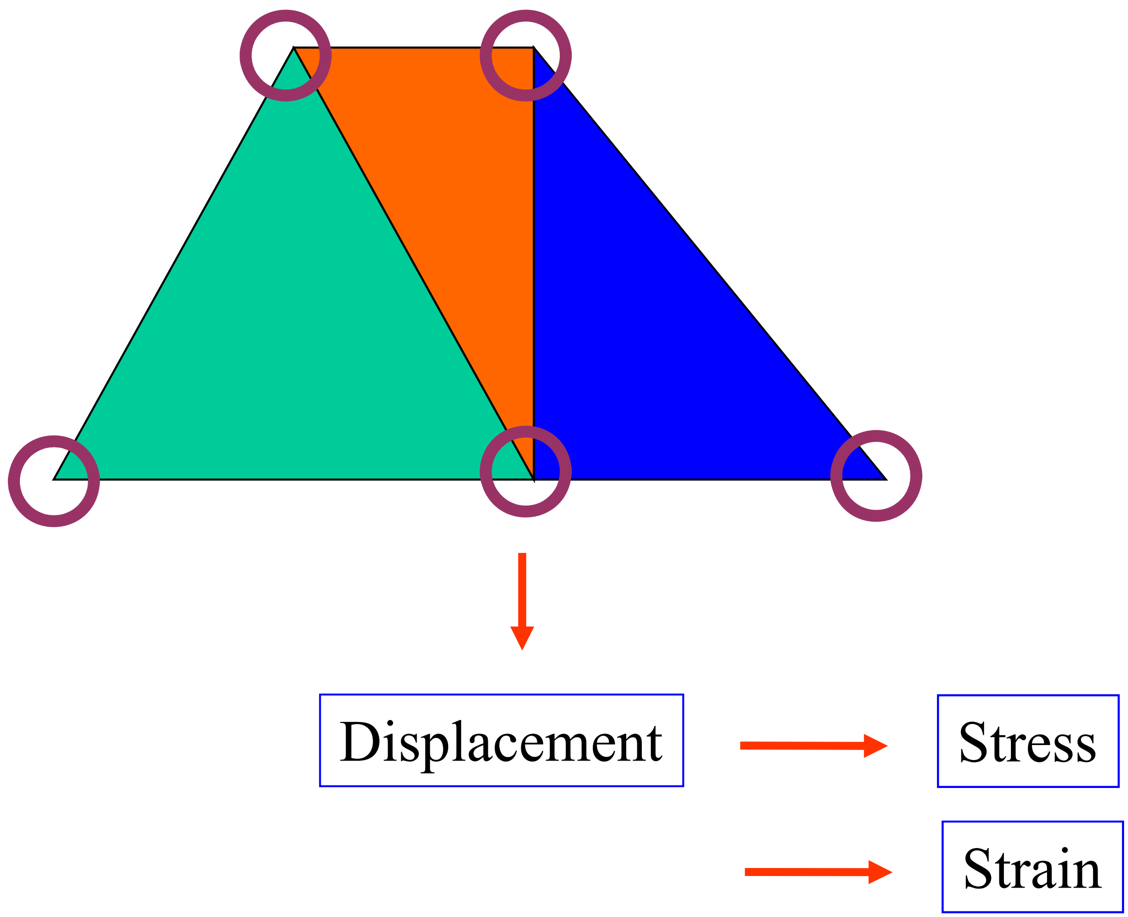
- **<u>Secondary quantities</u> (stresses and strains) are expressed in terms of nodal values of primary quantity**

Object

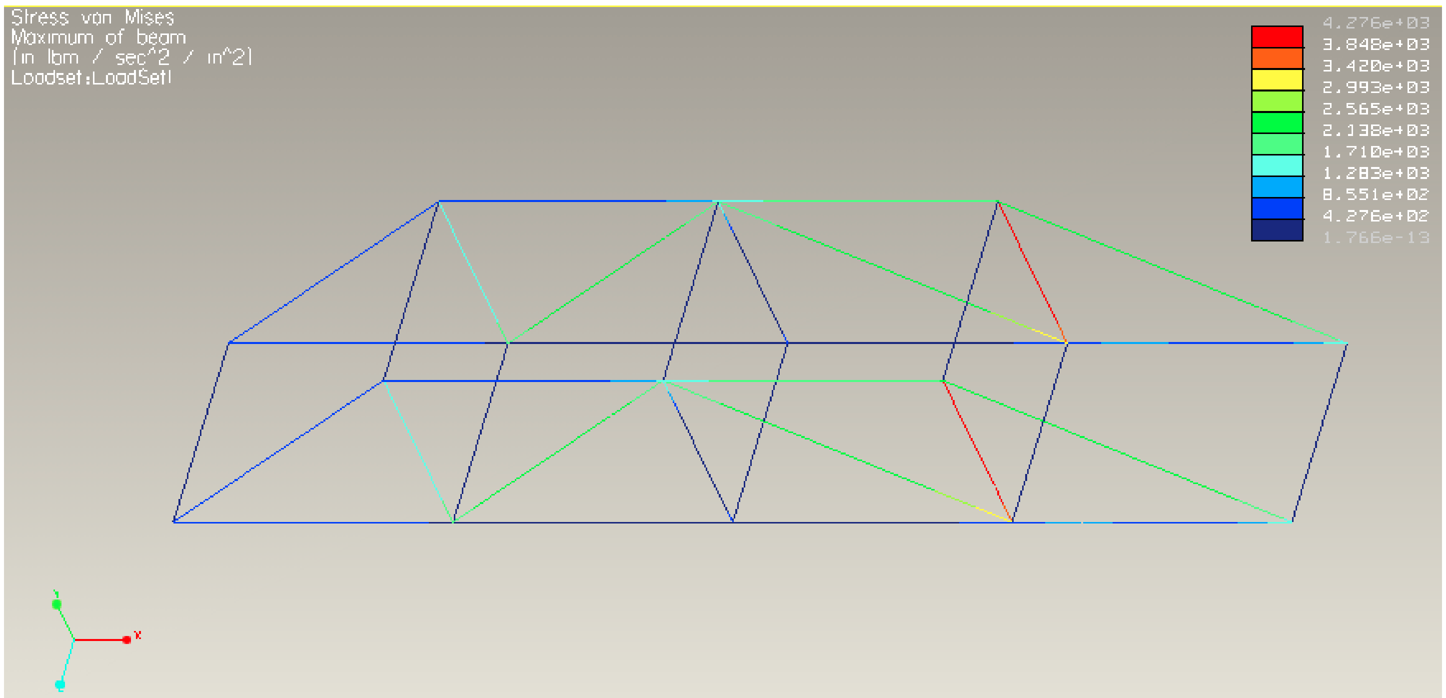
Elements



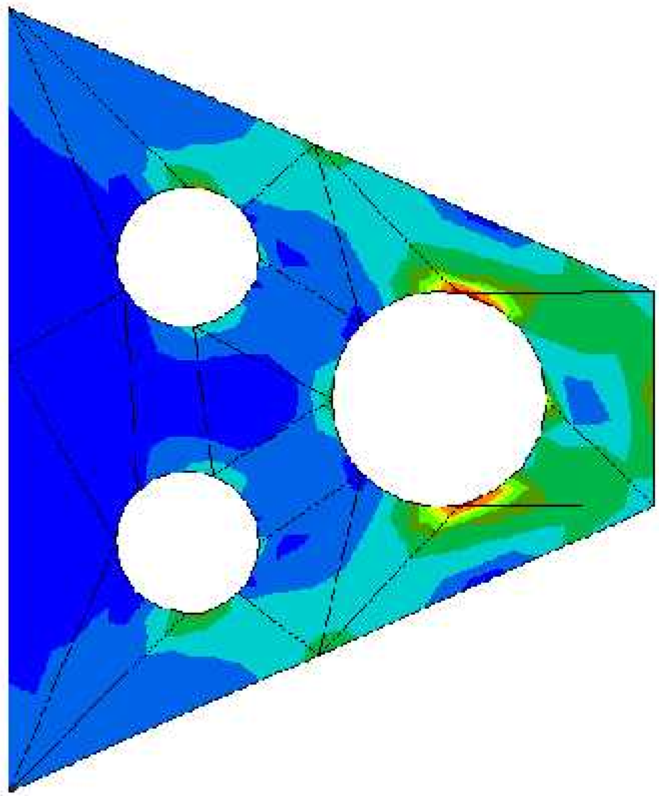
Nodes



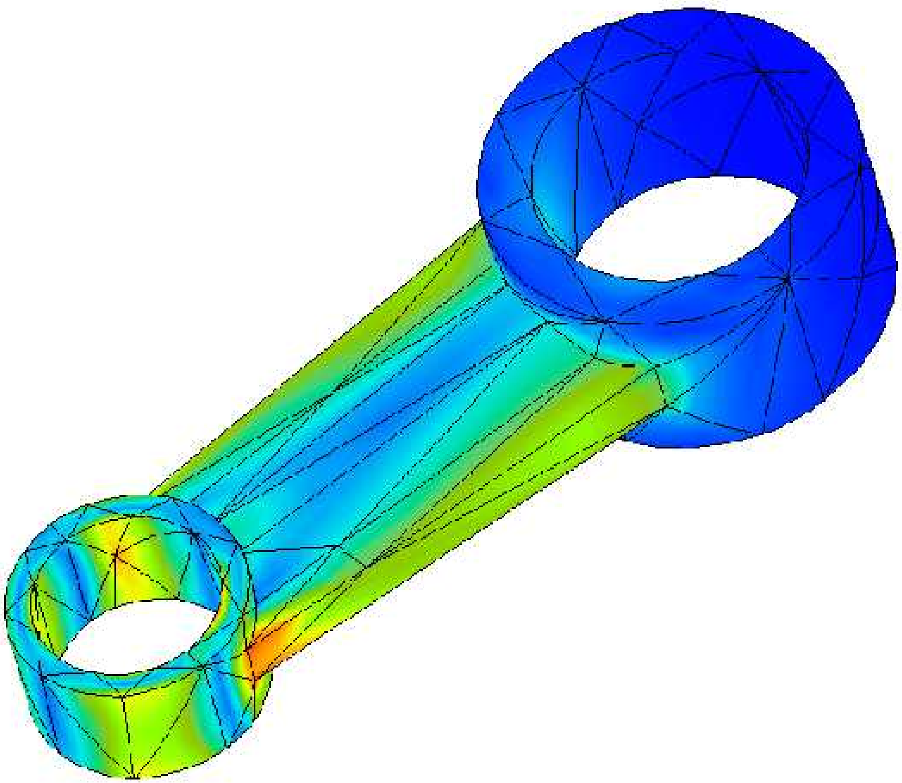
Examples of FEA – 1D (beams)



Examples of FEA - 2D



Examples of FEA – 3D



Advantages

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

The following notes are a summary from “Fundamentals of Finite Element Analysis” by David V. Hutton

Principles of FEA

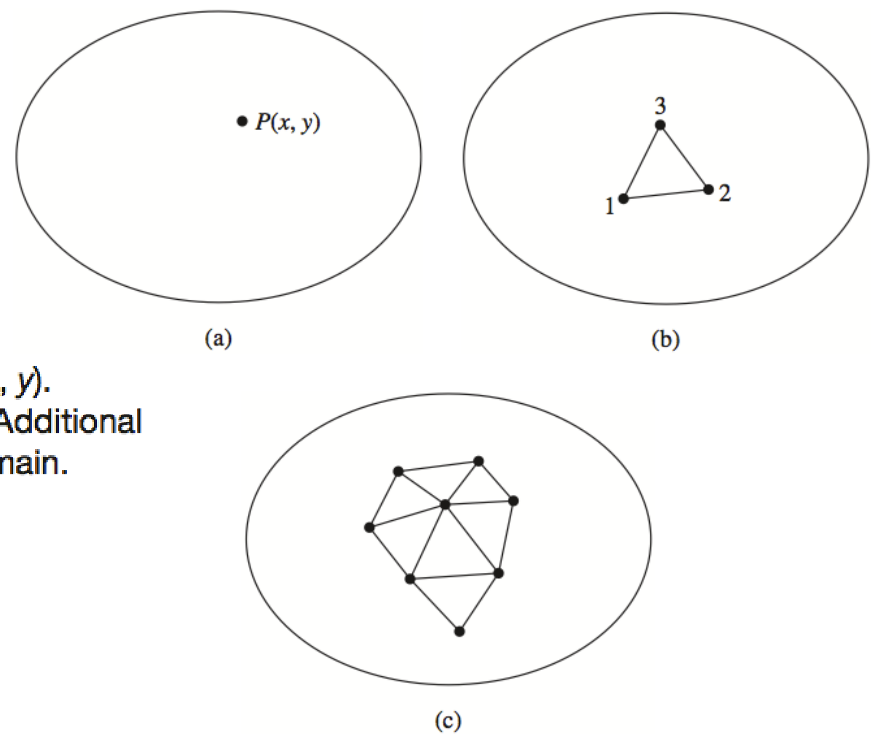
The finite element method (FEM), or finite element analysis (FEA), is a computational technique used to obtain ***approximate solutions of boundary value problems*** in engineering.

Boundary value problems are also called ***field problems***. The field is the domain of interest and most often represents a physical structure.

The ***field variables*** are the dependent variables of interest governed by the differential equation.

The *boundary conditions* are the specified values of the field variables (or related variables such as derivatives) on the boundaries of the field.

For simplicity, at this point, we assume a two-dimensional case with a single field variable $\varphi(x, y)$ to be determined at every point $P(x, y)$ such that a known governing equation (or equations) is satisfied exactly at every such point.



(a) A general two-dimensional domain of field variable $\phi(x, y)$. (b) A three-node finite element defined in the domain. (c) Additional elements showing a partial finite element mesh of the domain.

-A finite element is *not* a differential element of size dx × dy.

- A *node* is a specific point in the finite element at which the value of the field variable is to be explicitly calculated.

Shape Functions

The values of the field variable computed at the nodes are used to approximate the values at non-nodal points (that is, in the element interior) by *interpolation* of the nodal values. For the three-node triangle example, the field variable is described by the approximate relation

$$\varphi(x, y) = N_1(x, y)\,\varphi_1 + N_2(x, y)\,\varphi_2 + N_3(x, y)\,\varphi_3$$

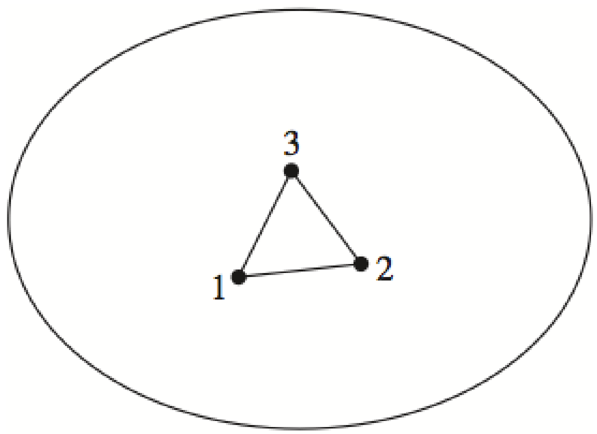
where φ_1, φ_2, and φ_3 are the values of the field variable at the nodes, and N_1, N_2, and N_3 are the ***interpolation functions,*** also known as ***shape functions*** **or** ***blending functions***.

In the finite element approach, the nodal values of the field variable are treated as unknown *constants* that are to be determined. The interpolation functions are most often polynomial forms of the independent variables, derived to satisfy certain required conditions at the nodes.

The interpolation functions are predetermined, *known* functions of the independent variables; and these functions describe the variation of the field variable within the finite element.

Degrees of Freedom

Again a two-dimensional case with a single field variable $\varphi(x, y)$. The triangular element described is said to have *3 degrees of freedom,* as three nodal values of the field variable are required to describe the field variable everywhere in the element (scalar).



$$\varphi(x, y) = N_1(x, y)\, \varphi_1 + N_2(x, y)\, \varphi_2 + N_3(x, y)\, \varphi_3$$

In general, the number of *degrees of freedom* associated with a finite element *is equal to the product of the number of nodes and the number of values of the field variable* (and possibly its derivatives) that must be computed at each node.

A GENERAL PROCEDURE FOR FINITE ELEMENT ANALYSIS

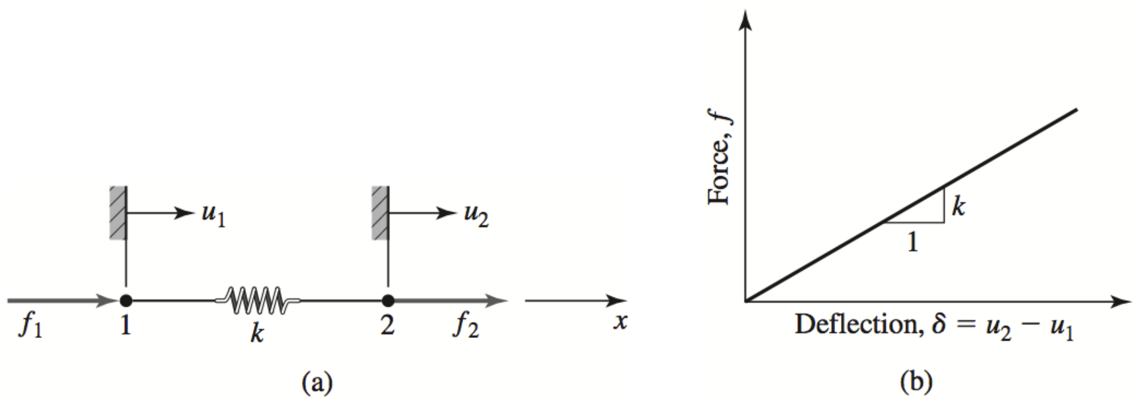
- **Preprocessing**
  - Define the geometric domain of the problem.
  - Define the element type(s) to be used (Chapter 6).
  - Define the material properties of the elements.
  - Define the geometric properties of the elements (length, area, and the like).
  - Define the element connectivities (mesh the model).
  - Define the physical constraints (boundary conditions). Define the loadings.
- **Solution**
  - computes the unknown values of the primary field variable(s)
  - computed values are then used by back substitution to compute additional, derived variables, such as reaction forces, element stresses, and heat flow.
- **Postprocessing**
  - Postprocessor software contains sophisticated routines used for sorting, printing, and plotting selected results from a finite element solution.

Stiffness Matrix

The primary characteristics of a finite element are embodied in the element ***stiffness matrix***. For a structural finite element, the stiffness matrix contains the geometric and material behavior information that indicates the resistance of the element to deformation when subjected to loading. Such deformation may include axial, bending, shear, and torsional effects. For finite elements used in nonstructural analyses, such as fluid flow and heat transfer, the term *stiffness matrix* is also used, since ***the matrix represents the resistance of the element to change when subjected to external influences***.

LINEAR SPRING AS A FINITE ELEMENT

A linear elastic spring is a mechanical device capable of supporting axial loading only, and the elongation or contraction of the spring is directly proportional to the applied axial load. The constant of proportionality between deformation and load is referred to as the *spring constant, spring rate,* or ***spring stiffness k***, and has units of force per unit length. As an elastic spring supports axial loading only, we select an *element coordinate system* (also known as a *local* coordinate system) as an x axis oriented along the length of the spring, as shown.



(a) Linear spring element with nodes, nodal displacements, and nodal forces.
(b) Load-deflection curve.

Assuming that both the nodal displacements are zero when the spring is undeformed, the net spring deformation is given by

$$\delta = u_2 - u_1$$

and the resultant axial force in the spring is

$$f = k\delta = k(u_2 - u_1)$$

For equilibrium,

$$f_1 + f_2 = 0 \quad \text{or} \quad f_1 = -f_2,$$

Then, in terms of the applied nodal forces as

$$\boldsymbol{f_1 = -k(u_2 - u_1)}$$

$$\boldsymbol{f_2 = k(u_2 - u_1)}$$

which can be expressed in matrix form as

$$\begin{bmatrix} k & -k \\ -k & k \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} f_1 \\ f_2 \end{Bmatrix} \qquad \text{or} \qquad [k_e]\{u\} = \{f\}$$

where

$$[k_e] = \begin{bmatrix} k & -k \\ -k & k \end{bmatrix}$$

Stiffness matrix for one spring element

is defined as the element stiffness matrix in the element coordinate system (or local system), $\{u\}$ is the column matrix (vector) of nodal displacements, and $\{f\}$ is the column matrix (vector) of element nodal forces.

$$\begin{Bmatrix} f_1 \\ f_2 \end{Bmatrix} = [k_e] \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} \qquad \text{with} \qquad [k_e] = \begin{bmatrix} k & -k \\ -k & k \end{bmatrix}$$

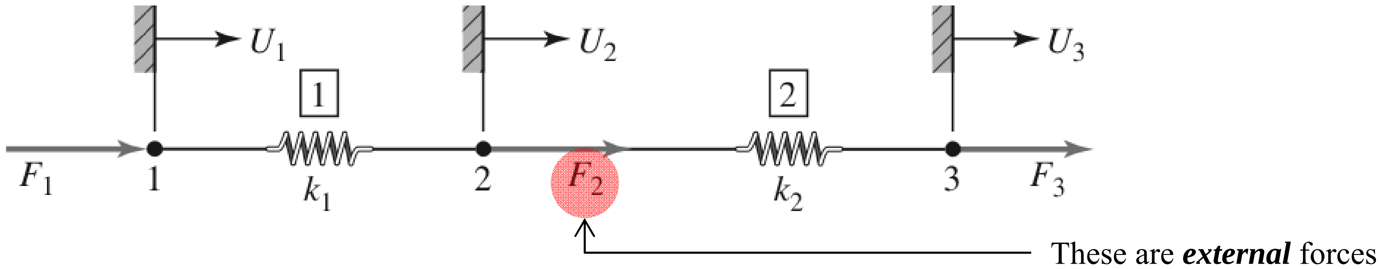
known $\{F\} = [K]\ \{X\}$ unknown

The equation shows that the element stiffness matrix for the linear spring element is a ***2 × 2 matrix***. This corresponds to the fact that the element exhibits ***two nodal displacements (or degrees of freedom)*** and that ***the two displacements are not independent (that is, the body is continuous and elastic)***.

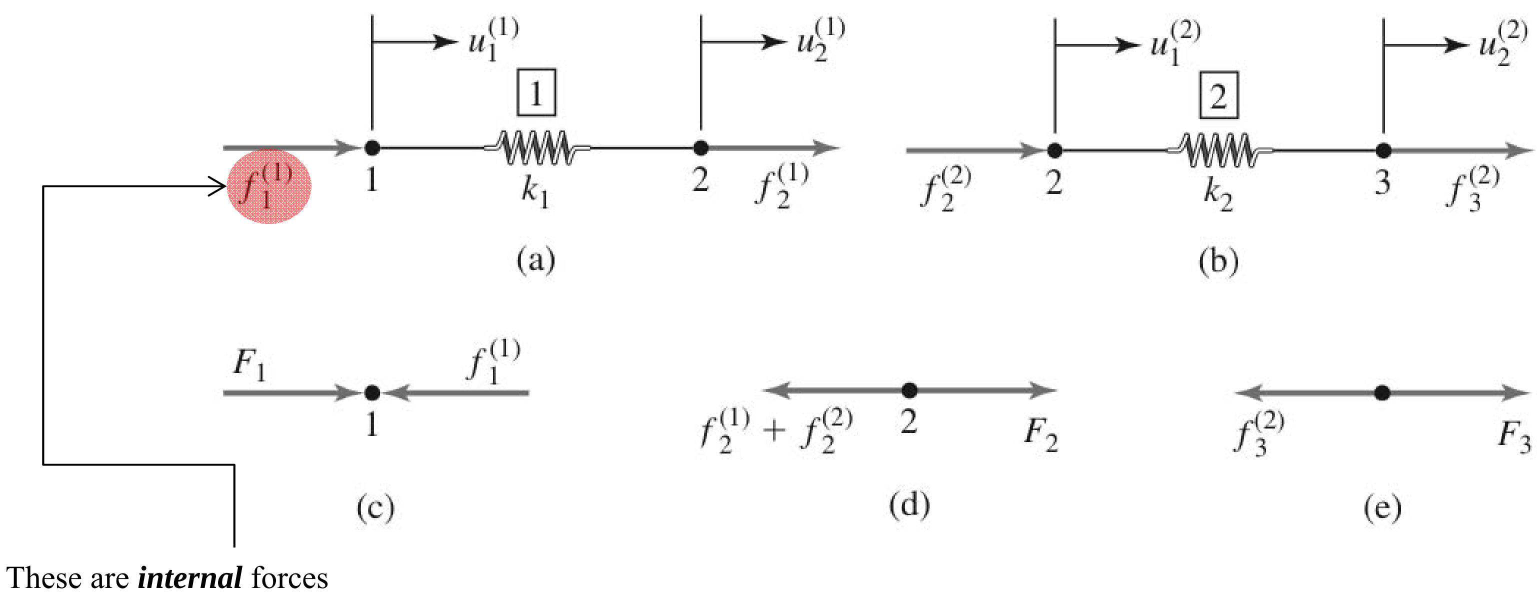
Furthermore, ***the matrix is symmetric***. This is a consequence of the symmetry of the forces (equal and opposite to ensure equilibrium).

Also ***the matrix is singular*** and therefore not invertible. That is because the problem as defined is incomplete and does not have a solution: ***boundary conditions are required***.

SYSTEM OF TWO SPRINGS



Free body diagrams:



Writing the equations for each spring in matrix form:

Superscript refers to element

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

$$\begin{bmatrix} k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_1^{(2)} \\ u_2^{(2)} \end{Bmatrix} = \begin{Bmatrix} f_2^{(2)} \\ f_3^{(2)} \end{Bmatrix}$$

To begin assembling the equilibrium equations describing the behavior of the system of two springs, the displacement ***compatibility conditions****,* which relate element displacements to system displacements, are written as:

$$u_1^{(1)} = U_1 \qquad u_2^{(1)} = U_2 \qquad u_1^{(2)} = U_2 \qquad u_2^{(2)} = U_3$$

And therefore:

$$\begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 \end{bmatrix} \begin{Bmatrix} U_1 \\ U_2 \end{Bmatrix} = \begin{Bmatrix} f_1^{(1)} \\ f_2^{(1)} \end{Bmatrix}$$

$$\begin{bmatrix} k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} U_2 \\ U_3 \end{Bmatrix} = \begin{Bmatrix} f_2^{(2)} \\ f_3^{(2)} \end{Bmatrix}$$

Here, we use the notation $f^{(j)}{}_i$ to represent the force exerted on element j at node i.

Expand each equation in matrix form:

$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 & 0 \\ 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} U_1 \\ U_2 \\ 0 \end{Bmatrix} = \begin{Bmatrix} f_1^{(1)} \\ f_2^{(1)} \\ 0 \end{Bmatrix}$$

$$\begin{bmatrix} 0 & 0 & 0 \\ 0 & k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} 0 \\ U_2 \\ U_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ f_2^{(2)} \\ f_3^{(2)} \end{Bmatrix}$$

Summing member by member:

$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 + k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} U_1 \\ U_2 \\ U_3 \end{Bmatrix} = \begin{Bmatrix} f_1^{(1)} \\ f_2^{(1)} + f_2^{(2)} \\ f_3^{(2)} \end{Bmatrix}$$

Next, we refer to the free-body diagrams of each of the three nodes:

$$f_1^{(1)} = F_1 \qquad f_2^{(1)} + f_2^{(2)} = F_2 \qquad f_3^{(2)} = F_3$$

Final form:

$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1+k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} U_1 \\ U_2 \\ U_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix} \qquad (1)$$

Where the stiffness matrix:

$$[K] = \begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1+k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix}$$

Note that the system stiffness matrix is:

(1) symmetric, as is the case with all linear systems referred to orthogonal coordinate systems;

(2) singular, since no constraints are applied to prevent rigid body motion of the system;

(3) the system matrix is simply ***a superposition of the individual element stiffness matrices*** with proper assignment of element nodal displacements and associated stiffness coefficients to system nodal displacements.

FEA for multiple (many) elements

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

Array of applied forces (one for each DOF)

Matrix of stiffnesses (DOF x DOF)

Array of displacements (one for each DOF)

$\{F\}$ is "known" (loads)

$[K]$ is "known" (geometry, material properties…elements)

$\{U\}$ is to be determined (displacements)

This can be solved mathematically using a matrix inversion method

$$\{F\} = [K] \cdot \{U\} \quad \rightarrow \quad \{U\} = [K]^{-1}\{F\}$$

(first nodal quantity)

Once the displacements $\{U\}$ are known, then strains and stresses can be determined:

$\varepsilon = \dfrac{\Delta u}{L}$ (1-D …more complicated for 2-D and 3-D strains)

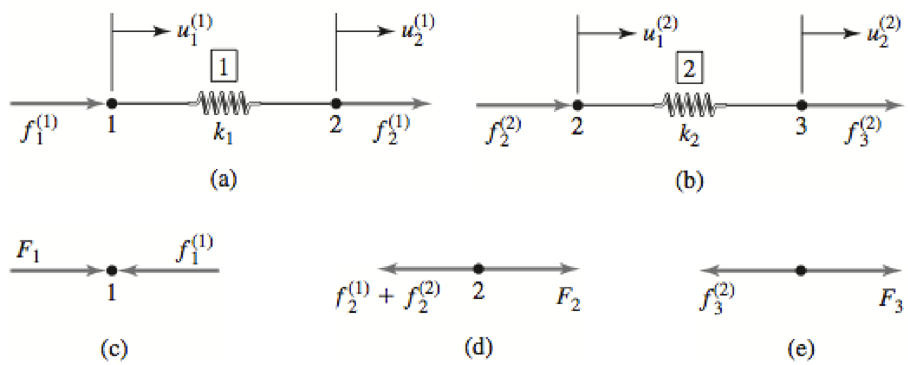
$\sigma = E \cdot \varepsilon$

and $FOS = \dfrac{\sigma_y}{\sigma}$

(second nodal quantities)

Example with Boundary Conditions

Consider the two element system as described before where Node 1 is attached to a fixed support, yielding the displacement constraint $U_1 = 0$, $k_1 = 50$ lb/in, $k_2 = 75$ lb/in, $F_2 = F_3 = 75$ lb for these conditions determine nodal displacements U_2 and U_3.



Substituting the specified values into (1) we have:

Due to boundary condition

$$\begin{bmatrix} 50 & -50 & 0 \\ -50 & 125 & -75 \\ 0 & -75 & 75 \end{bmatrix} \begin{Bmatrix} 0 \\ U_2 \\ U_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ 75 \\ 75 \end{Bmatrix}$$

Example with Boundary Conditions

Because of the constraint of zero displacement at node 1, nodal force F_1 becomes an unknown reaction force. Formally, the first algebraic equation represented in this matrix equation becomes:

$$-50U_2 = F_1$$

and this is known as a ***constraint equation**,* as it represents the equilibrium condition of a node at which the displacement is constrained. The second and third equations become

$$\begin{bmatrix} 125 & -75 \\ -75 & 75 \end{bmatrix} \begin{Bmatrix} U_2 \\ U_3 \end{Bmatrix} = \begin{Bmatrix} 75 \\ 75 \end{Bmatrix}$$

which can be solved to obtain $U_2 = 3$ in. and $U_3 = 4$ in. *Note that the matrix equations governing the unknown displacements are obtained by simply striking out the first row and column of the 3 × 3 matrix system, since the constrained displacement is zero (homogeneous)*. If the displacement boundary condition is not equal to zero (nonhomogeneous) then this is not possible and the matrices need to be manipulated differently (partitioning).

Truss Element

The spring element is also often used to represent the elastic nature of supports for more complicated systems. A more generally applicable, yet similar, element is an elastic bar subjected to axial forces only. This element, which we simply call a *bar or truss element,* is particularly useful in the analysis of both two- and three-dimensional frame or truss structures. Formulation of the finite element characteristics of an elastic bar element is based on the following assumptions:

1. The bar is geometrically straight.
2. The material obeys Hooke's law.
3. Forces are applied only at the ends of the bar.
4. The bar supports axial loading only; bending, torsion, and shear are not transmitted to the element via the nature of its connections to other elements.

Truss Element Stiffness Matrix

Let's obtain an expression for the stiffness matrix K for the beam element. Recall from elementary strength of materials that the deflection δ of an elastic bar of length L and uniform cross-sectional area A when subjected to axial load P :

$$\delta = \frac{PL}{AE}$$

where E is the modulus of elasticity of the material. Then the equivalent spring constant k:

$$k = \frac{P}{\delta} = \frac{AE}{L}$$

Therefore the stiffness matrix for one element is:

$$[k_e] = \frac{AE}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

And the equilibrium equation in matrix form:

$$\frac{AE}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} f_1 \\ f_2 \end{Bmatrix}$$

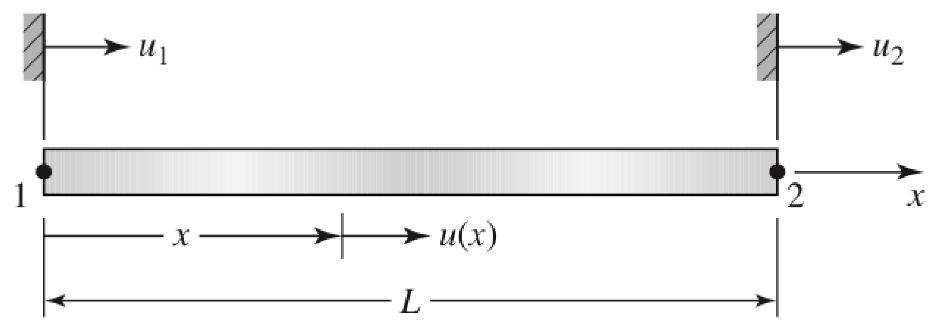
Truss Element Blending Function

An elastic bar of length L to which is affixed a uniaxial coordinate system x with its origin arbitrarily placed at the left end. This is the *element* coordinate system or reference frame. Denoting axial displacement at any position along the length of the bar as $u(x)$, we define nodes 1 and 2 at each end as shown and introduce the nodal displacements:

$u_1 = u\ (x=0)$ and $u_2 = u(x = L)$.

Thus, we have the continuous field variable $u(x)$, which is to be expressed (approximately) in terms of two nodal variables u_1 and u_2. To accomplish this discretization, we assume the existence of *interpolation* functions $N_1(x)$ and $N_2(x)$ (also known as *shape* or *blending* functions) such that

$$u(x) = N_1(x)u_1 + N_2(x)u_2$$



Truss Element Blending Function

To determine the interpolation functions, we require that the boundary values of $u(x)$ (the nodal displacements) be identically satisfied by the discretization such that:

$$u_1 = u\ (x=0) \text{ and } u_2 = u(x = L).$$

lead to the following boundary (nodal) conditions:

$$\begin{aligned} N_1(0) &= 1 \quad N_2(0) = 0 \\ N_1(L) &= 0 \quad N_2(L) = 1 \end{aligned}$$

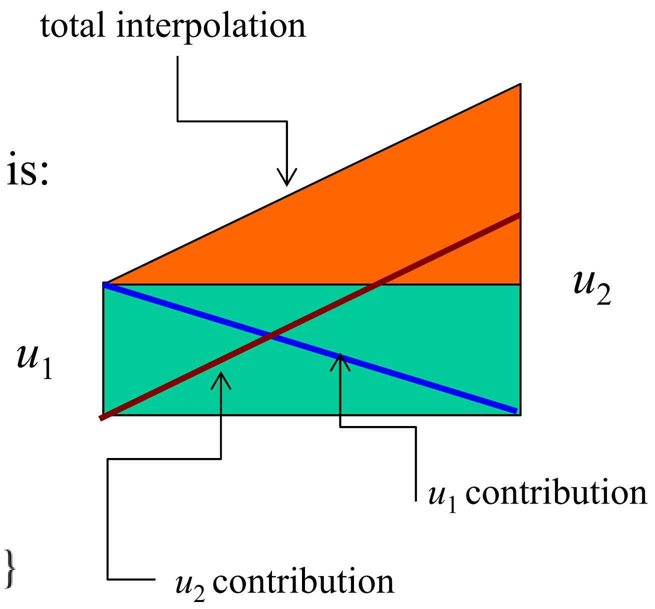
As we have two conditions that must be satisfied by each of two one-dimensional functions, the simplest forms for the interpolation functions are polynomial forms:

$$\begin{aligned} N_1(x) &= a_0 + a_1 x \\ N_2(x) &= b_0 + b_1 x \end{aligned}$$

Truss Element Blending Function

Where the polynomial coefficients are to be determined via satisfaction of the boundary (nodal) conditions. Application of conditions yields $a_0 = 1$, $b_0 = 0$, therefore $a_1 = -(1/L)$ and $b_1 = x/L$. Therefore, the interpolation functions are

$$N_1(x) = 1 - x/L$$
$$N_2(x) = x/L$$



Therefore the final expression of the blending function is:

$$u(x) = (1 - x/L)u_1 + (x/L)u_2$$

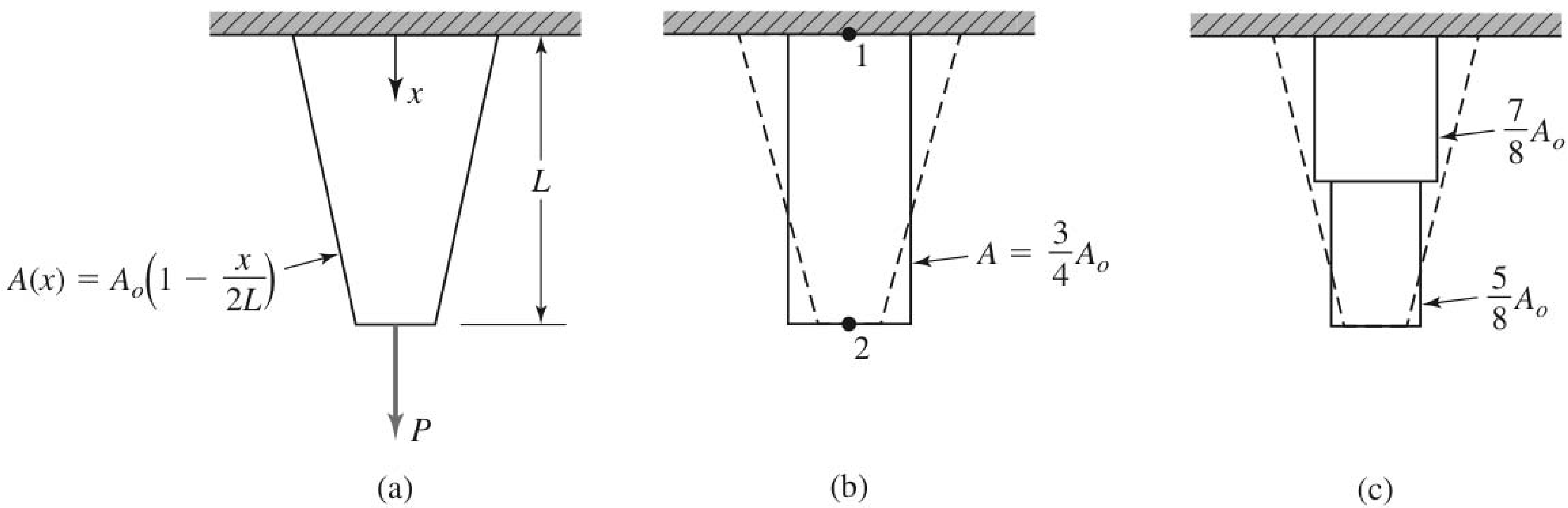
And in matrix form:

$$u(x) = [N_1(x) \quad N_2(x)]\begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = [N]\{u\}$$

This is the *displacement field* in terms of *nodal variables*.

Truss Element Example

Figure depicts a tapered elastic bar subjected to an applied tensile load P at one end and attached to a fixed support at the other end. The cross-sectional area varies linearly from A_0 at the fixed support at $x = 0$ to $A_0/2$ at $x = L$. Calculate the displacement of the end of the bar (a) by modeling the bar as a single element having cross-sectional area equal to the area of the actual bar at its midpoint along the length, (b) using two bar elements of equal length and similarly evaluating the area at the midpoint of each, and compare to the exact solution.



Truss Element Example Solution a)

For a single element, the cross-sectional area is $3A_0/4$ and the element "spring constant" and element equation are:

$$k = \frac{AE}{L} = \frac{3A_0 E}{4L} \qquad \text{and} \qquad \frac{3A_0 E}{4L}\begin{bmatrix} 1 & -1 \\ -1 & -1 \end{bmatrix}\begin{Bmatrix} U_1 \\ U_2 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ P \end{Bmatrix}$$

Applying the constraint condition $U_1 = 0$, we find for U_2 as the displacement at $x = L$

$$U_2 = \frac{4PL}{3A_0 E} = 1.333\,\frac{PL}{A_0 E}$$

Truss Element Example Solution b)

Two elements of equal length $L/2$ with associated nodal displacements. For element 1, $A_1 = 7A_0/8$ so

$$k_1 = \frac{A_1 E}{L_1} = \frac{7A_0 E}{8(L/2)} = \frac{7A_0 E}{4L}$$

while for element 2, we have

$$A_1 = \frac{5A_0}{8} \quad \text{and} \quad k_2 = \frac{A_2 E}{L_2} = \frac{5A_0 E}{8(L/2)} = \frac{5A_0 E}{4L}$$

Since no load is applied at the center of the bar, the equilibrium equations for the system of two elements is:

$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 + k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} U_1 \\ U_2 \\ U_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ 0 \\ P \end{Bmatrix}$$

Applying the constraint condition $U_1 = 0$ results in

$$\begin{bmatrix} k_1 + k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} U_2 \\ U_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ P \end{Bmatrix}$$

Truss Element Example Solution b)

Adding the two equations gives

$$U_2 = \frac{P}{k_1} = \frac{4PL}{7A_0E}$$

and substituting this result into the first equation results in

$$U_3 = \frac{k_1 + k_2}{k_2} = \frac{48PL}{35A_0E} = 1.371\frac{PL}{A_0E}$$

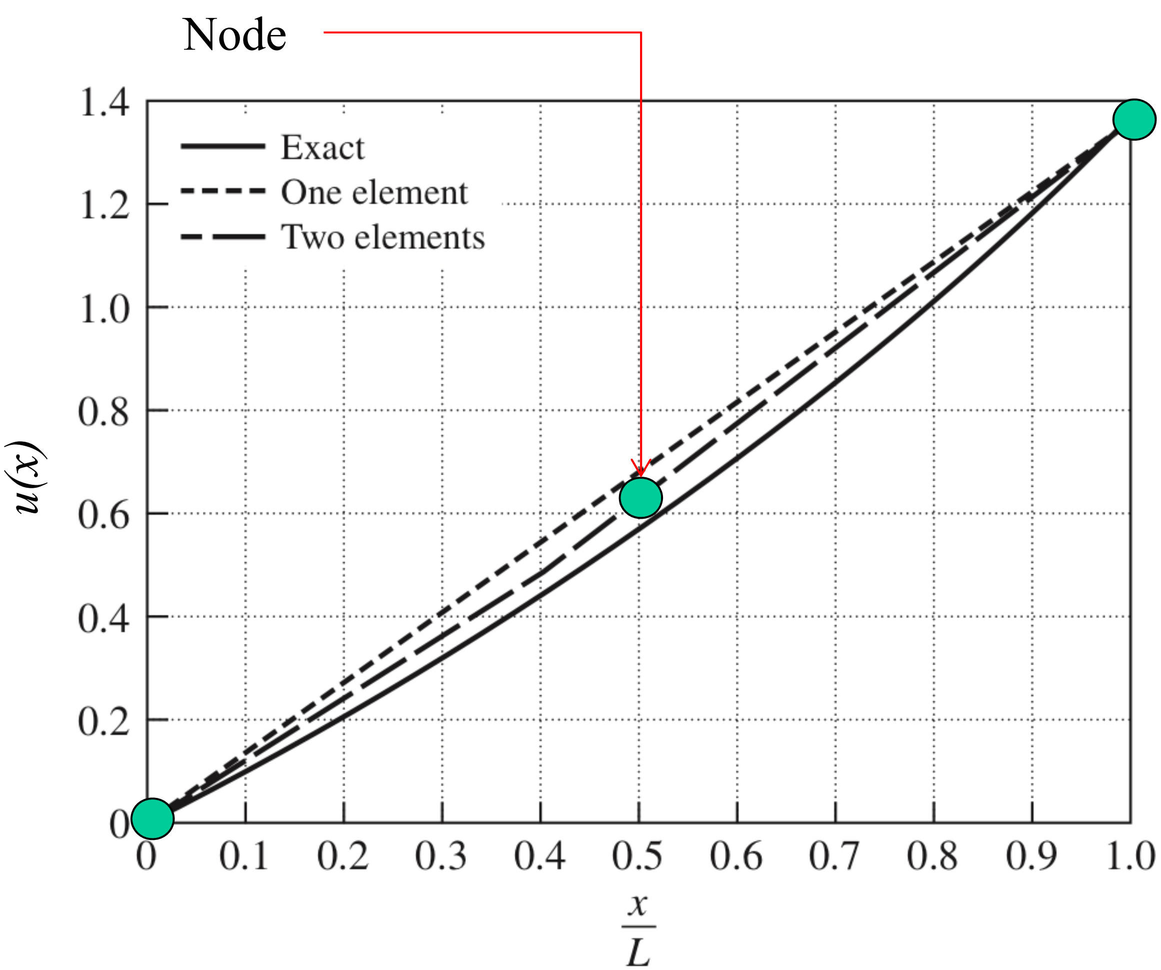
Comparing the displacement for the three solution at $x = L$:

a) $1.333\dfrac{PL}{A_0E}$

b) $1.371\dfrac{PL}{A_0E}$

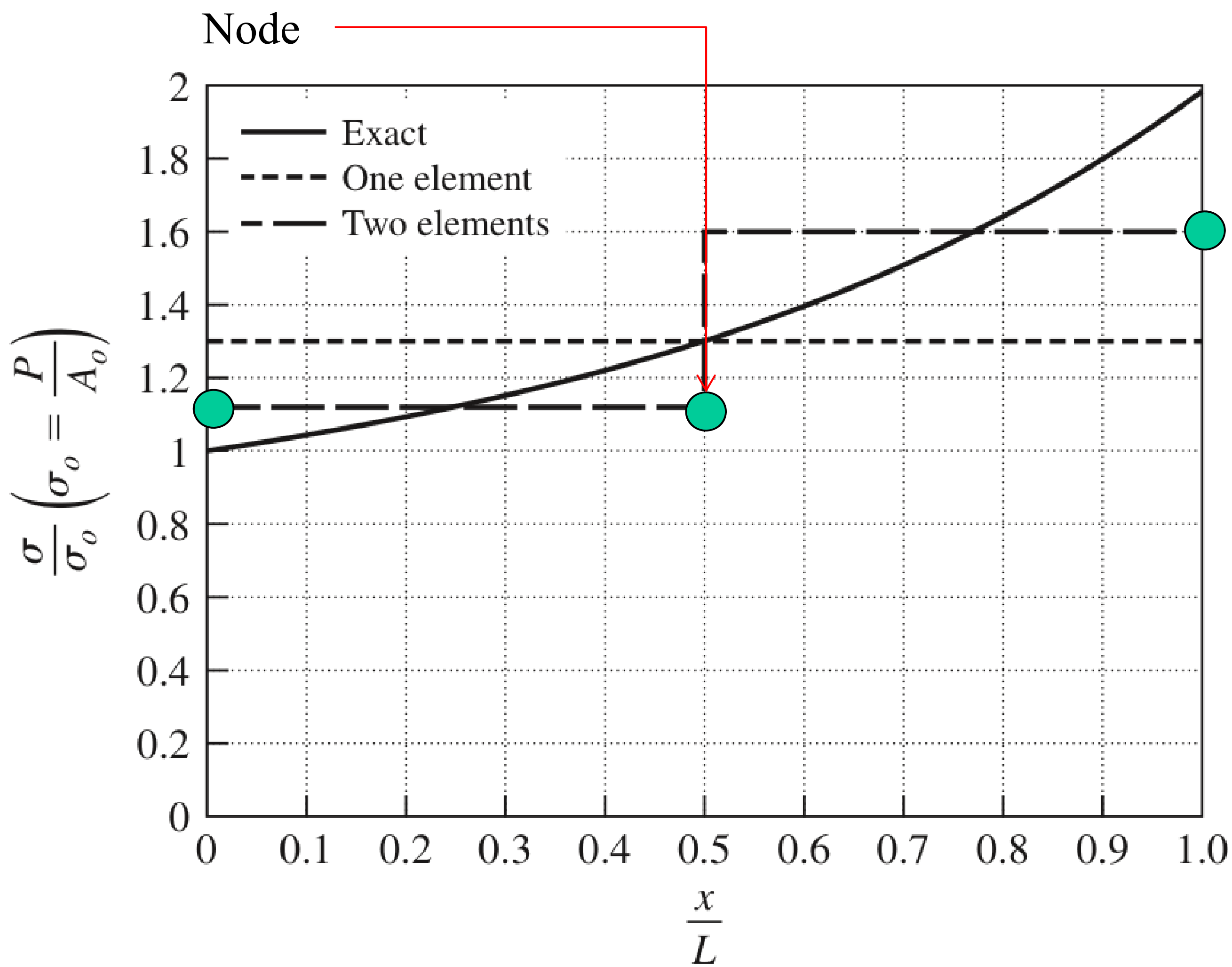
c) Exact solution $1.386\dfrac{PL}{A_0E}$

Truss Element Example Solution Comparison Deflection



Shape function for interpolated values: $u(x) = (1 - x/L)u_1 + (x/L)u_2$

Truss Element Example Solution Comparison Stress



For stress results are much different, discontinuous for FEA and highly dependent on number of nodes

Beam Element

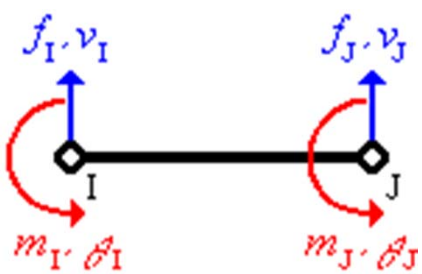
The usual assumptions of elementary beam theory are applicable here:

1. The beam is loaded only in the y direction.

2. Deflections of the beam are small in comparison to the characteristic dimensions of the beam.

3. The material of the beam is linearly elastic, isotropic, and homogeneous. The beam is prismatic and the cross section has an axis of symmetry in the plane of bending.

Beam Element



$$\underbrace{\begin{Bmatrix} f_I \\ m_I \\ f_J \\ m_J \end{Bmatrix}}_{4 \times 1} = \overbrace{\underbrace{\begin{bmatrix} k_{11} & k_{12} & k_{13} & k_{14} \\ k_{21} & k_{22} & k_{23} & k_{24} \\ k_{31} & k_{32} & k_{33} & k_{34} \\ k_{41} & k_{42} & k_{43} & k_{44} \end{bmatrix}}_{4 \times 4}}^{\text{stiffness matrix}} \underbrace{\begin{Bmatrix} v_I \\ \theta_I \\ v_J \\ \theta_J \end{Bmatrix}}_{4 \times 1}$$

The equation shows that the element stiffness matrix for the beam element is a ***4 × 4 matrix***. This corresponds to the fact that the element exhibits ***four degrees of freedom*** and that ***the displacements are not independent (that is, the body is continuous and elastic)***.

Furthermore, ***the matrix is symmetric***. This is a consequence of the symmetry of the forces and moments (equal and opposite to ensure equilibrium).

Also ***the matrix is singular*** and therefore not invertible. That is because the problem as defined is incomplete and does not have a solution: ***boundary conditions are required***.

Beam Element Shape Function and Stiffness Matrix

Shape function:

$$v(x) = f(v_1, v_2, \theta_1, \theta_2, x)$$

$$v(x) = (1 - 3\xi^2 + 2\xi^3)v_1 + L(\xi - 2\xi^2 + \xi^3)\theta_1 + (3\xi^2 - 2\xi^3)v_2 + L\xi^2(\xi - 1)\theta_2$$

With $\xi = \dfrac{x}{L}$

And the Stiffness Matrix:

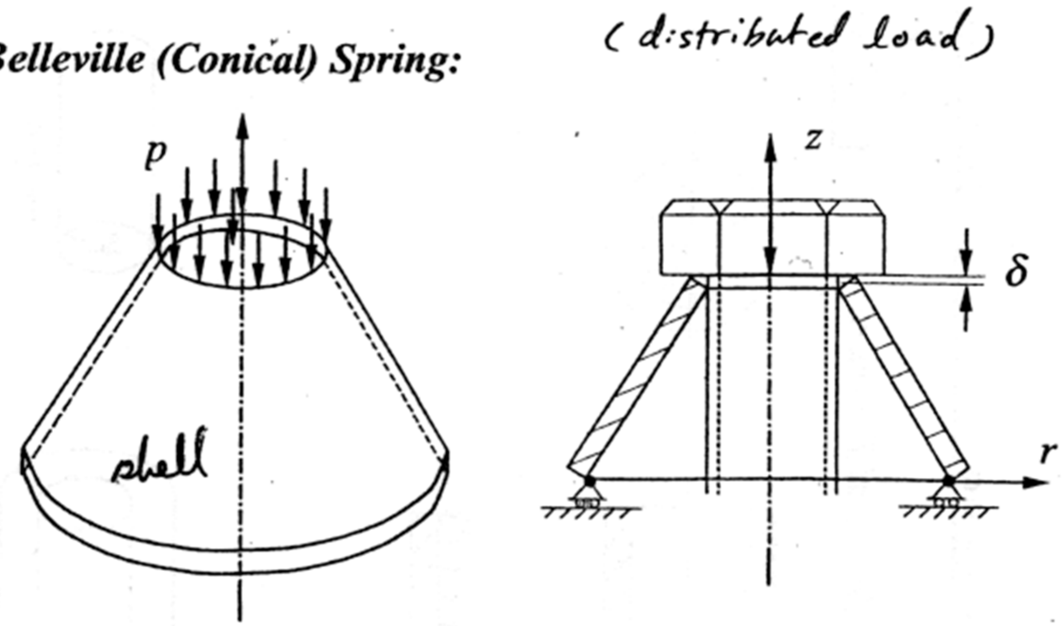
$$[k_e] = \frac{EI_z}{L^3}\begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix}$$

Way of <u>Stacking</u> Blocks/Elements

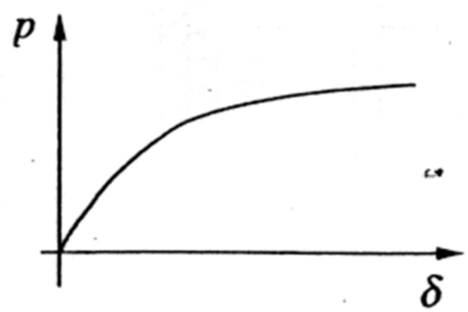
- **Compatibility requirement: ensures that the "displacements" at the shared node of adjacent elements are <u>equal</u>.**
- **Equilibrium requirement: ensures that elemental forces and the external forces applied to the system nodes are in <u>equilibrium</u>.**
- **Boundary conditions: ensures the system satisfy the boundary constraints and so on.**

Limitations of Regular FEA Software

- **Belleville (Conical) Spring:**



- Unable to handle geometrically nonlinear - large deformation problems: shells, rubber, etc.



This is a geometrically nonlinear (large deformation) problem and iteration method (incremental approach) needs to be employed.

Interpolation Functions for General Element Formulation

In finite element analysis, solution accuracy is judged in terms of convergence as the element "mesh" is refined.
There are two major methods of mesh refinement.

In the first, known as ***h-refinement****,* mesh refinement refers to the process of increasing the number of elements used to model a given domain, consequently, reducing individual element size.

In the second method, ***p-refinement****,* element size is unchanged but the order of the polynomials used as interpolation functions is increased.

The objective of mesh refinement in either method is to obtain sequential solutions that exhibit asymptotic convergence to values representing the exact solution.