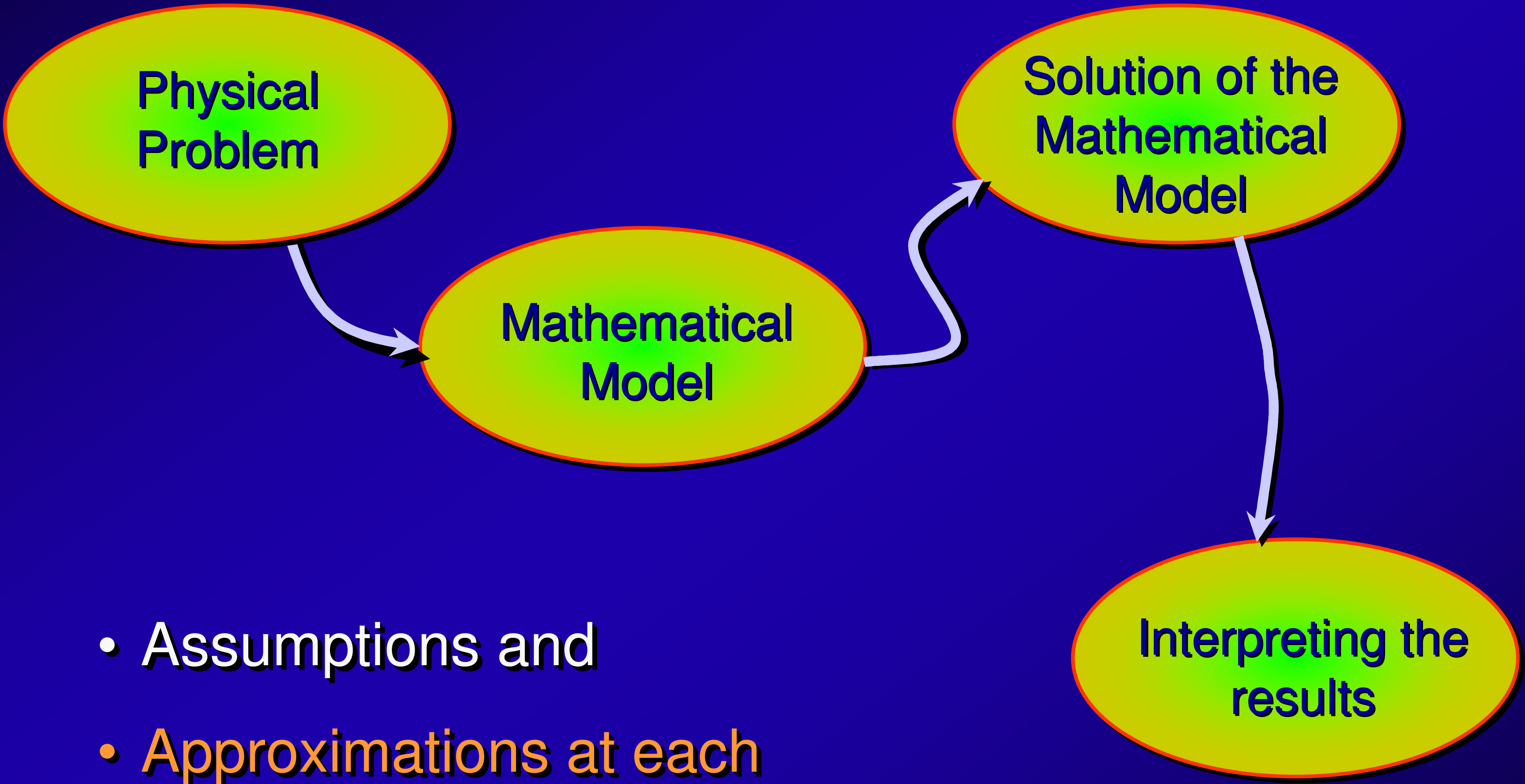
The
Finite Element Method

Mohammed Ameen, PhD
Department of Civil Engineering

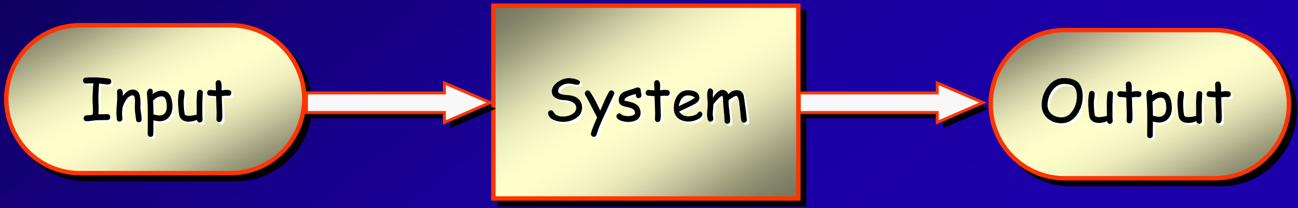
National Institute of Technology Calicut

Introduction



- Assumptions and
- Approximations at each of the above stages

Analysis, Synthesis & Calibration



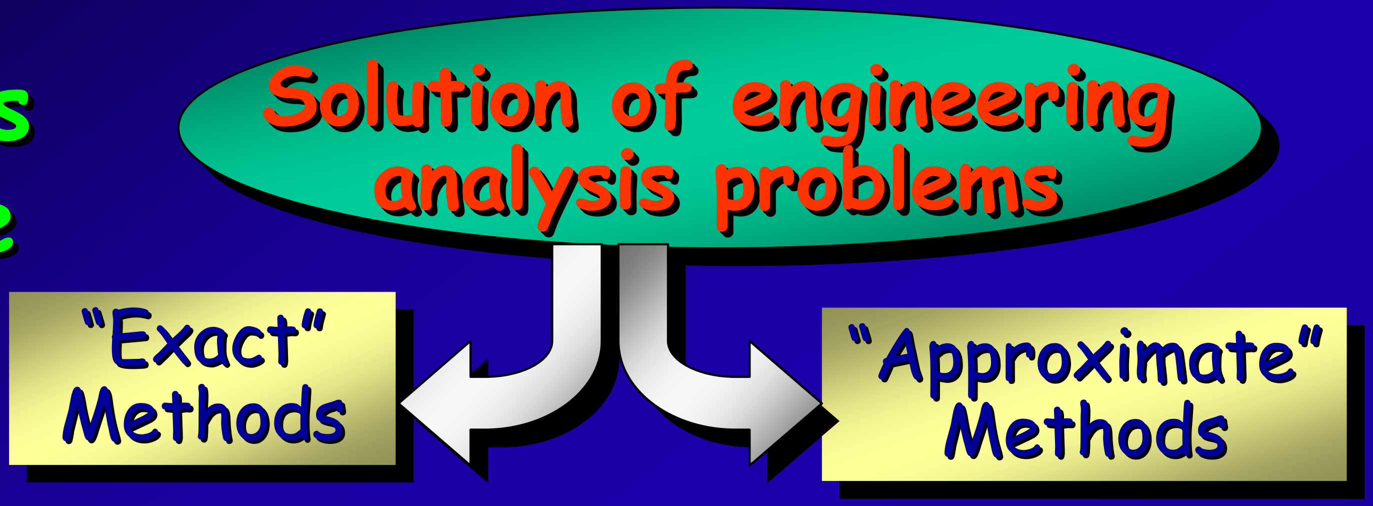
Three types of associated problems:

Given the System and the Input: Find the Output— “Analysis”. Direct problem; unique solution (for linear system).

Given the Input and the Output: Find the System parameters—“Synthesis” (or Design). Inverse problem; nonunique solution; Scope for optimisation.

Given the System and the Output: Find the Input—“Calibration”. Inverse problem; ill-conditioned equations; regularisation.

Exact versus Approximate Methods



Solution of engineering analysis problems

"Exact" Methods

"Approximate" Methods

Exact methods:– Analytical methods based on theory of differential equations (or integral equations, integro-differential equations, etc.)

Uses many assumptions to reduce the problem to a simple one and solves this simple problem "exactly"

Mainly of academic interest only. Often acts as a bench mark solution to test approximate methods

Approximate methods:– Numerical methods based on some approximation theory – uses computer oriented numerical methods to solve "exact" problem "approximately" (but with a high accuracy); practically very useful.

Approximate Methods

Among various approximate methods the following are the most popular ones:

- Finite Element Method (FEM)
- Finite Difference Method (FDM)
- Boundary Element Method (BEM)

The first two are classified under “domain” methods and the third is classified as a “boundary” method

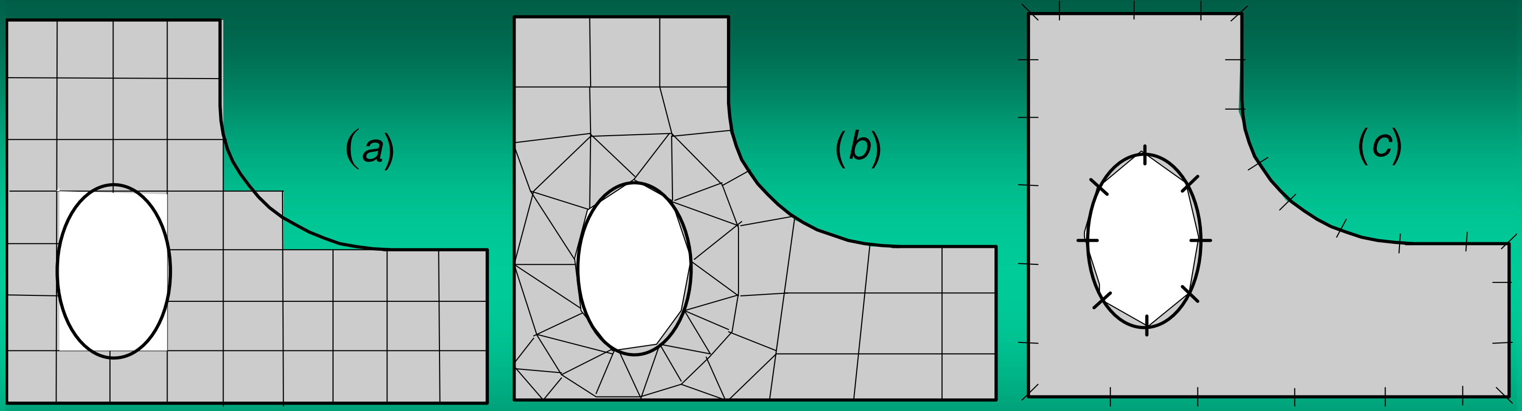


Figure (*a*) Finite difference, (*b*) Finite element and (*c*) Boundary element discretisations of a machine component.

The Finite Element Method

Finite element method is one of the most popular and versatile *approximate methods* used for solving real-life engineering problems

Today it is used to analyse problems such as:

- Stress analysis
- Heat transfer
- Fluid flow
- Lubrication
- Electric and magnetic fields
- Piezoelectricity
- and many many others

- Problems which were intractable are now being solved routinely
- Finite element procedures are used in the design of:
    - Buildings
    - Electric motors
    - Heat engines
    - Ships
    - Aircrafts/spacecrafts
- Manufacturing companies and big design offices have one or more large in-house finite element programs

A Brief History of FEM

1906 – Lattice analogy to solve continuum problems in which the continuum was replaced by a regular mesh of elastic bars

1941 – Applications of lattice analogy in plane elasticity and plate bending problems

1941 – Courant suggested piecewise polynomial approximations over triangular subregions as a way to get approximate numerical solutions

Early to Mid 1950's – Engineers in aeronautical industry made remarkable progress; e.g. Turner (US) devised three-noded triangular element to model the wing skin, Taig did similar work in UK, Argyris in Germany included FE concepts in a series of papers on matrix methods

1956 – Turner, Clough, Martin & Topp – Classic paper

1960 – Clough coined the name “Finite Element Method”

1963 – FEM acquired respectability in academia when it was recognised as a form of Rayleigh-Ritz method

1965 – Papers about heat conduction and seepage flow

1966 – Isoparametric elements

1967 –The first text book on FE by Zienkiewicz and Cheung

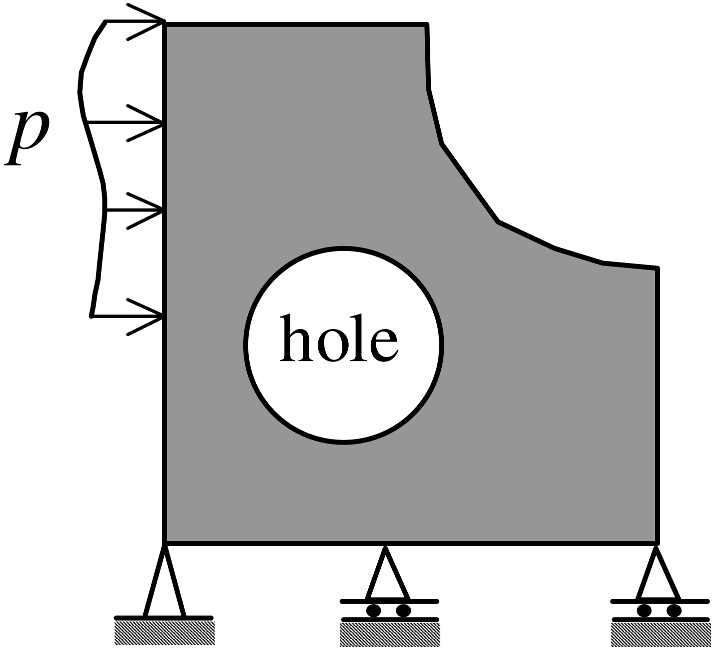
In 1961 – 10 papers about FEM were published

In 1966 – 134 papers;

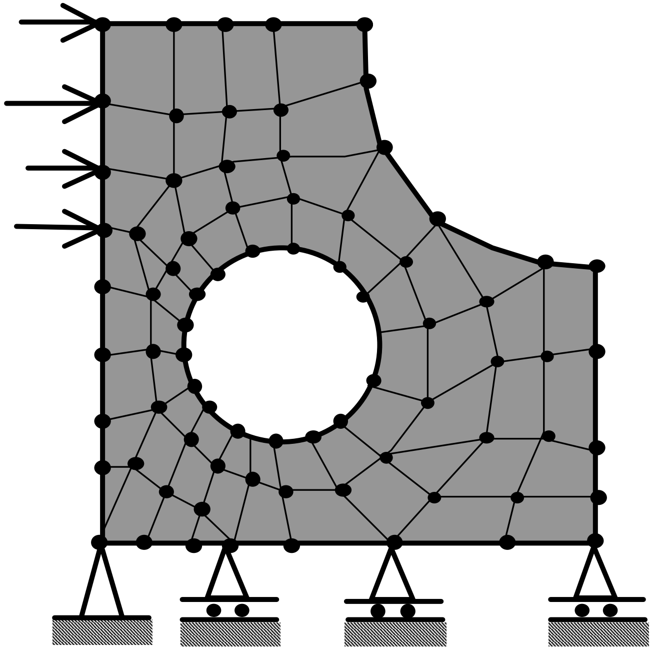
In 1971 – 844 papers

By 1986 – there were more than 20,000 papers!

In 1995 – Mackerle estimated that about 3800 papers on FEM were being published annually, and that the cumulative total of FEM publications amounted to some 380 books, 400 conference proceedings and 56,000 papers, and 310 general purpose FE computer programs!



A machine component

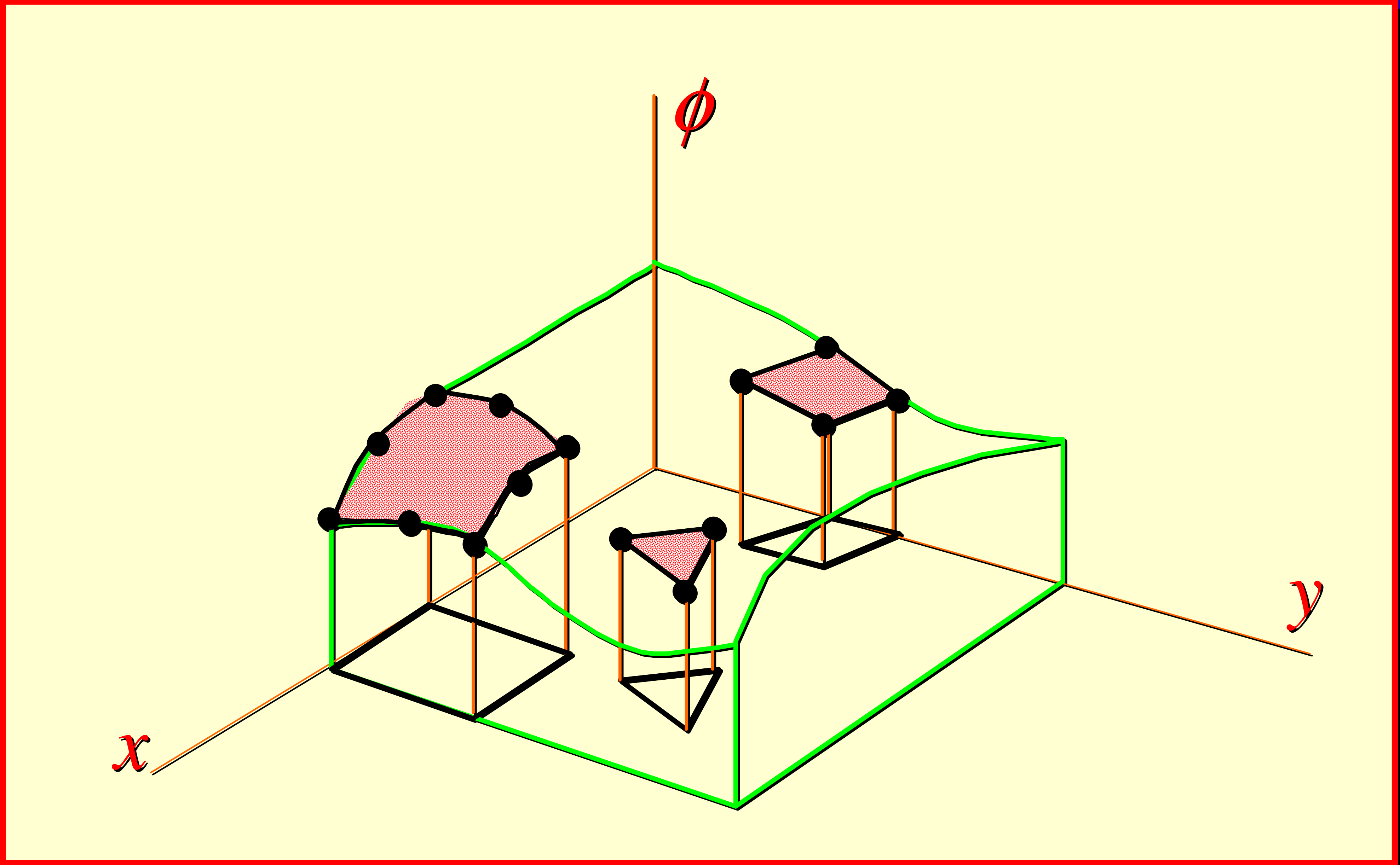


Finite element discretisation

The displacements and stresses caused by the pressure *p* are required

- The domain is discretised into finite elements which are connected to each other only at nodes
- Each element is of simpler geometry – hence easier to analyse than the actual structure
- A complicated solution is approximated by piece-wise continuous simple solutions
- Total number of degrees of freedom (d.o.f) = $d\,n$ (d = number of d.o.f. per node; n = number of nodes)

- Algebraic equations that describe the finite element model are generated and solved to determine the d.o.f.
- Sawing the continuum into pieces and then pinning the pieces together → gaps and overlaps at inter-element boundaries – "inter-element compatibility"
- However, between elements there may be jumps in the *x* and *y* derivatives of ϕ



Piece-wise continuous simple solutions to approximate the actual field of ϕ

Some Useful References

Books:

- R.D. Cook, D.S. Malkus, M.E. Plesha, and R.J. Witt, *Concepts and Applications of Finite Element Analysis*, John Wiley, New York, 2002. (An excellent book to both beginners and users of FEM, very well written)
- J.K. Bathe, *Finite Element Procedures in Engineering Analysis*, Prentice Hall of India, New Delhi, 1992. (A very good reference book especially for problems related to structural dynamics. An exhaustive book, which is a bit hard to read at places where even the notations are complicated)

- J.K. Bathe and E.L. Wilson, *Numerical Methods in Finite Element Analysis*, Prentice Hall India, New Delhi, 1987. (A much smaller book in comparison to the above; contains almost all materials related to dynamics as in the above)
- O.C. Zienkiewicz, and R.L. Taylor, *The Finite Element Method*, Volume I & II McGraw-Hill, London, 1989. (An excellent reference book. Volume II covers advanced topics such as nonlinear problems)
- O.C. Zienkiewicz, and K. Morgan, *Finite Elements and Approximation*, John Wiley, New York, 1983. (A good book on weighted residual approach)

- B. Szabo and I. Babuska, *Finite Element Analysis*, John Wiley, New York, 1991. (For a good taste in more mathematical treatment of FEM)
- S.S. Rao, *The Finite Element Method in Engineering*, Pergamon Press, New York, 1982.
- Shames and C.L. Dym, *Energy and Finite Element Methods in Structural Mechanics*, New Age Intrnl-Wiley Eastern, New Delhi, 1991.

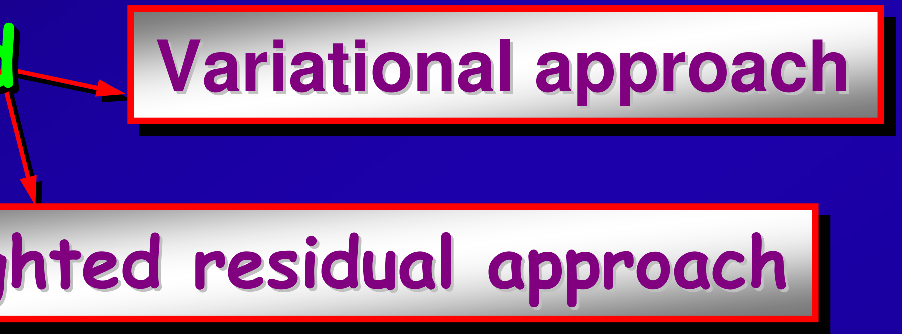
Web Resources:

Plenty of information available on web.

See for example:

http://homepage.usask.ca/~ijm451/finite/fe_resources/

The Finite Element Method



Variational approach

Weighted residual approach

Weighted residual method: The residue is obtained by replacing the unknown variable(s) in the governing differential equation of the problem by an approximate solution.

The residue is “weighted” and then minimised to arrive at an approximate solution – more versatile as this method can be used to solve any problem whose governing differential equation is known.

Variational method: based on natural variational principles. The solution is obtained by minimising the functional. The first finite element applications were based on this approach.

The Weighted Residual Method

Let the governing differential equation be:

$\mathcal{L}(u) = b$ in V + associated boundary conditions

e.g:

$$\mathcal{L}(\,) \equiv \frac{d^4}{dx^4}(\,) + \frac{d}{dx}(\,) + (\,)$$

Or

$$\mathcal{L}(\,) \equiv \frac{\partial^2}{\partial x^2}(\,) + \frac{\partial^2}{\partial y^2}(\,)$$

[2D Laplace's operator

Let $\hat{u}$ be an approximate solution: $\hat{u} = \sum_{i=1}^{n} \alpha_i \, \phi_i$

where α_i – undetermined parameters

and ϕ_i – linearly independent functions taken from a complete sequence of functions such as $\phi_1(x),\ \phi_2(x),\ \ldots\ ,\ \phi_n(x)$

Then, the “residual” or “error” function is:

$$R \equiv \mathcal{L}(\hat{u}) - b \neq 0 \text{ in } V$$

In the WRM’s, these errors are forced to zero in certain average sense.

Some of the common WRM’s are:

- Collocation method (point collocation)
- Collocation by subregion
- Galerkin method

The weighted residual statement is:

$$\int_V R\, w\, dV = 0 \qquad \text{(A)}$$

where $w = \beta_1\psi_1 + \beta_2\psi_2 + \cdots + \beta_k\psi_k$, is called the weighing function.

In the Galerkin method, ψ_i are chosen as ϕ_i itself.

Weak Formulations

(A) Leads to: $$\int_V w\,\{\mathcal{L}(\hat{u}) - b\}\; dV = 0 \qquad \text{(B)}$$

Integrating (B) by parts, we get:

$$\int_V w\,\{\mathcal{L}(\hat{u}) - b\}\; dV = \int_V \{\mathcal{L}_1(w)\mathcal{L}_2(\hat{u}) - w\,b\}\; dV + \int_S \cdots dS = 0$$

The above is said to be in the *weak form*.

E.g:– Let $\mathcal{L}(u) - b \equiv \frac{d^2u}{dx^2} + u + x = 0$ in $0 < x < 1$

The weighted residual statement is:

$$\int_0^1 \{\frac{d^2u}{dx^2} + u + x\} w\, dx = 0$$

which leads to:

$$-\int_0^1 \frac{du}{dx}\frac{dw}{dx} dx + \int_0^1 (u+x) w\, dx + w\frac{du}{dx}\bigg|_0^1 = 0$$

The above is in the “*weak form*”.

- The continuity requirements of the weak form are less rigid.
- The Finite Element formulations are based on the weak form.
- The main advantage of the weighted residual formulation is that if the governing differential equation of the problem is known, one can proceed to obtain an approximate solution.
- The weak form can be integrated by parts again to arrive at the inverse form.
- The Boundary Element Method (BEM) stems from the inverse form.

Example:
Beam problem

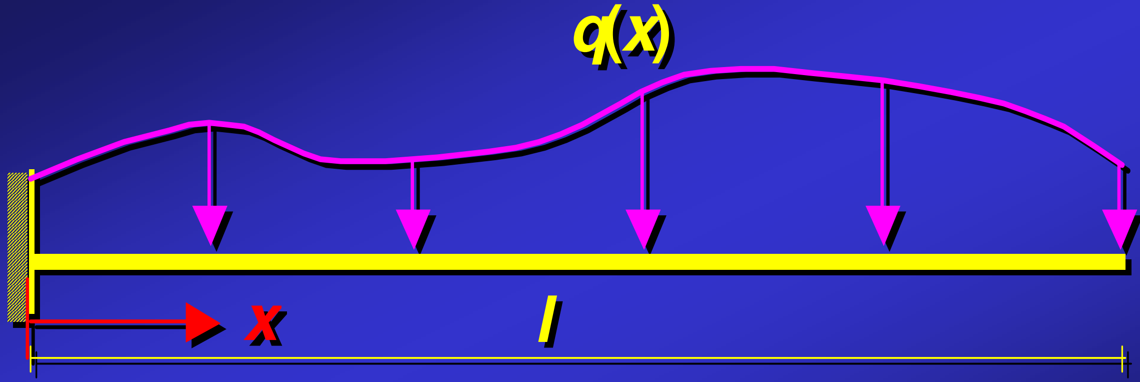


Figure Cantilever beam subject to loading

Differential equation approach:

Governing differential equation and associated boundary conditions are:

$$EI\frac{d^4w}{dx^4}=q(x);\quad \underbrace{w(0)=\left.\frac{dw}{dx}\right|_{(0)}=0;}_{\text{geometric b.c's (essential b.c's)}}\quad \underbrace{\left.\frac{d^2w}{dx^2}\right|_{(l)}=\left.\frac{d^3w}{dx^3}\right|_{(l)}=0}_{\text{natural b.c's (non-essential b.c's)}}$$

Variational Approach
(Principle of minimum potential energy)

The potential energy of the cantilever beam is given by:

$$\Pi_p = \int_0^l \frac{1}{2} EI(w'')^2 dx - \int_0^l q(x)w(x)dx$$

The principle of stationary potential energy:

"Among all the admissible configurations of a conservative system, those that satisfy the equations of equilibrium make the potential energy stationary w.r.t. small admissible variations of displacement"

Applying a small variation to $w(x)$ in the above, we get:

$$\delta \Pi_p = \int_0^l EIw'' \delta w'' dx - \int_0^l q \, \delta w \, dx = 0$$

Integrating the first term on the right by parts, we get:

$$\delta\Pi_p = EIw''\delta w'\Big|_0^l - EIw'''\delta w\Big|_0^l + \int_0^l EIw''''.\delta w.dx - \int_0^l q.\delta w.dx = 0$$

$$\Rightarrow \quad \delta\Pi_p = EIw''\delta w'\Big|_0^l - EIw'''\delta w\Big|_0^l + \int_0^l (EIw''''-q)\delta w.dx = 0$$

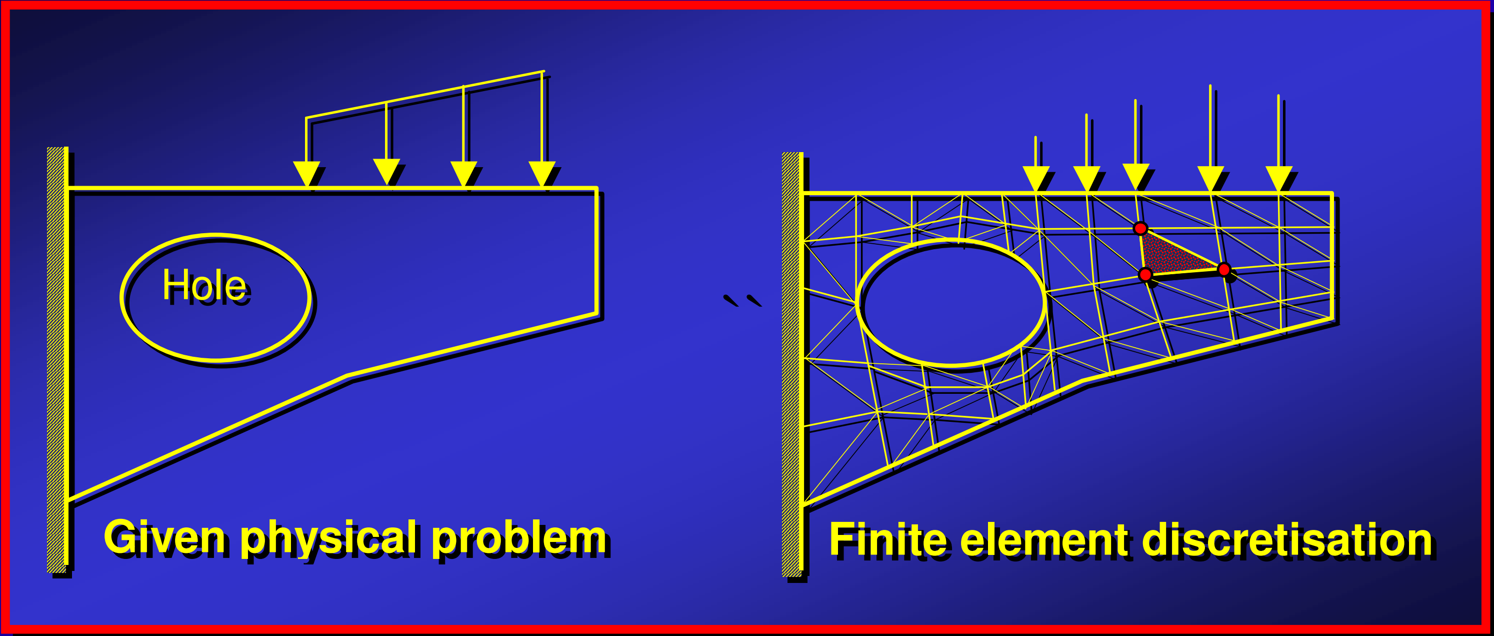
Since $\delta w = \delta w' = 0$ at $x = 0$, the above leads to the following:

$EI\, w'''' = q(x)$ [Euler-Bernaulli equation

and $EI\, w'' (x = l) = 0$ [Natural boundary conditions

$EI\, w'''(x = l) = 0$ [" " "

Steps involved in FE analysis



Discretise the continuum into many subregions called *finite elements* of arbitrary size, shape and orientation

Each element is assumed to be connected to the neighbouring elements only at a finite number of discrete points called *nodes*

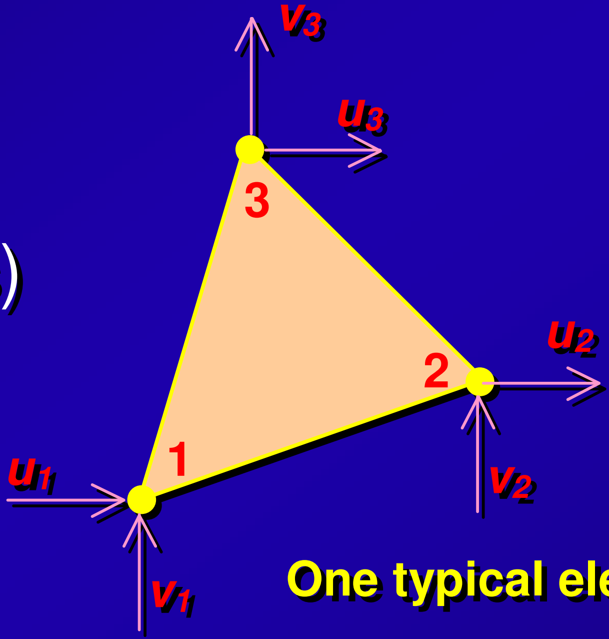
The displacements at the nodes are assumed as the basic unknowns of the problem

Thus, the number of degrees of freedom of the problem is reduced from infinity to a finite number

Concept of Interpolation

Now, let us assume (for the time being) that by some means we know the nodal degrees of freedom (say, displacements) of any one typical finite element;

Can we then calculate the displacement field within the element?



One typical element

$$\begin{Bmatrix} u(x,y) \\ v(z,y) \end{Bmatrix} = [N(x,y)]\{u^e\}$$

Or
$$\begin{Bmatrix} u(x,y) \\ v(z,y) \end{Bmatrix} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{Bmatrix}$$

$N_i(x,y)$ are called the *interpolation functions* or *shape functions*

Example:1

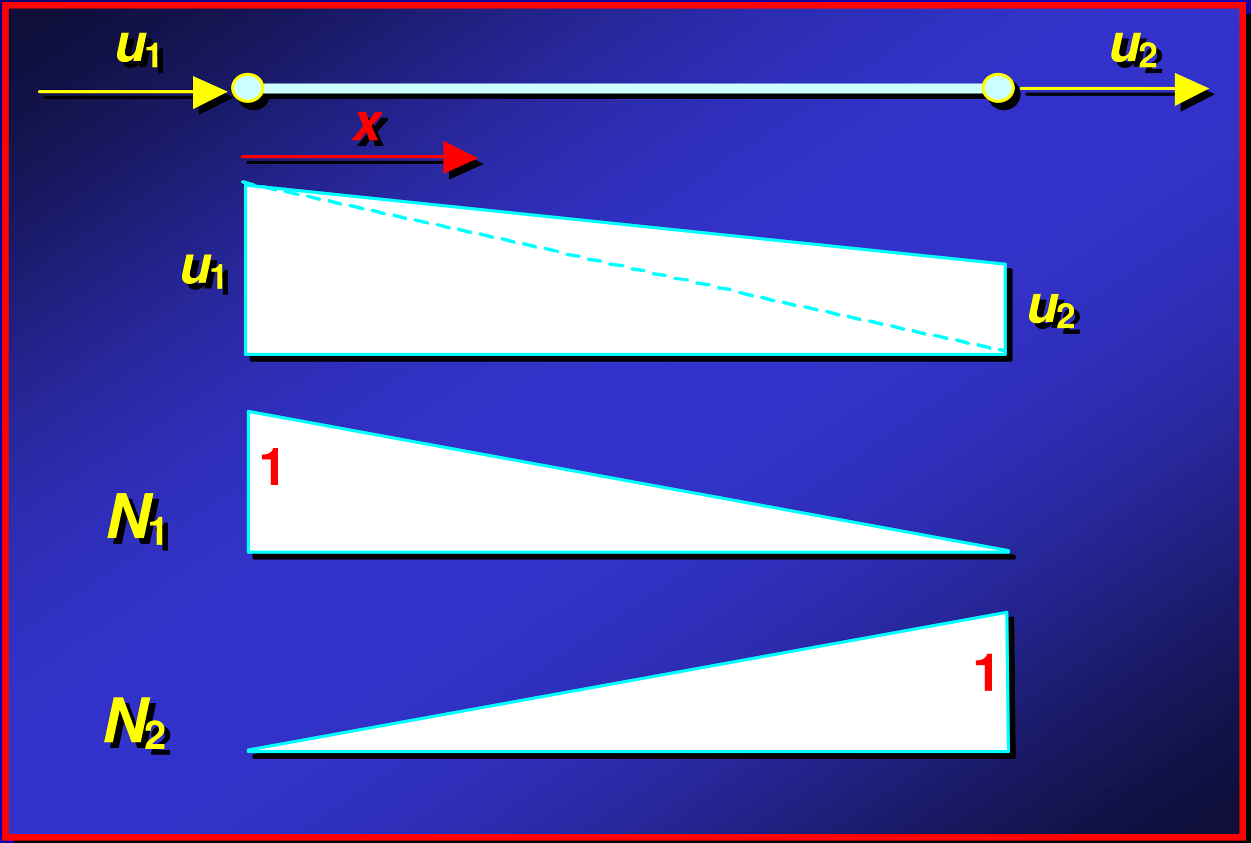
Bar element

$$u(x) = a + bx$$

$$@\ x = 0,\ u = u_1$$

$$@\ x = l,\ u = u_2$$

$$u(x) = (1 - x/l)\ u_1 + (x/l)\ u_2 = N_1\ u_1 + N_2\ u_2$$



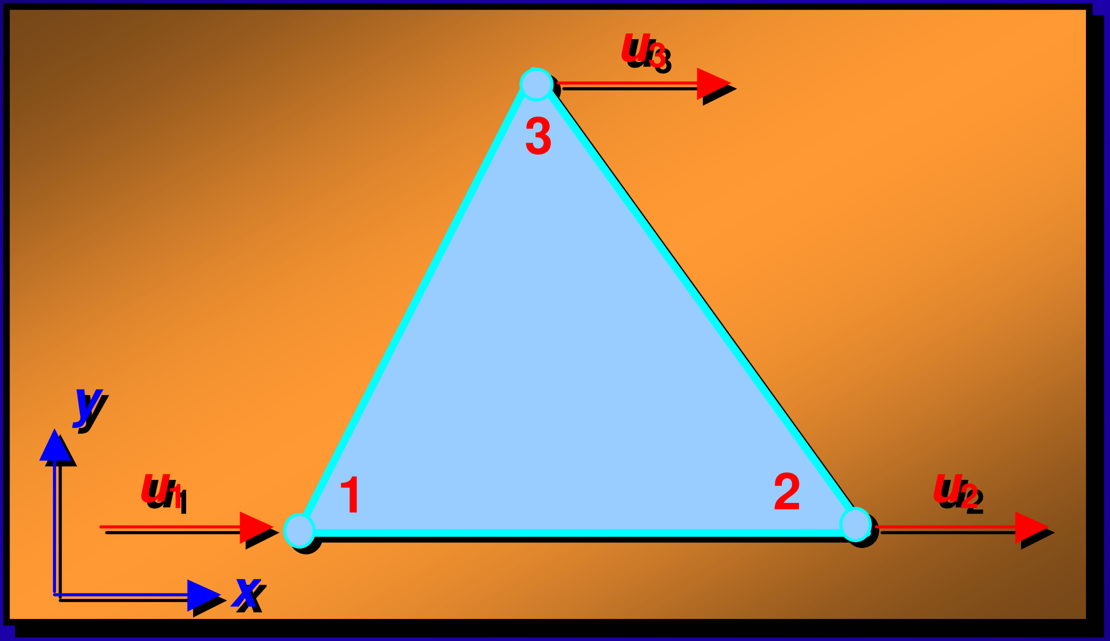
Thus N_1, N_2 are one-dimensional interpolation functions given by:

$$N_1(x) = (1 - x/l) \text{ and } N_2(x) = x/l$$

Example:2
Triangular plane element

$u(x, y)$ can be written as:

$$u(x,y) = \mathrm{a} + \mathrm{b}\,x + \mathrm{c}\,y$$
$$= [1 \quad x \quad y]\begin{Bmatrix} a \\ b \\ c \end{Bmatrix}$$
$$= [x]\{a\}$$



a, b, c are determined by solving the following system of equations:

$$\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{bmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \end{bmatrix}\begin{Bmatrix} a \\ b \\ c \end{Bmatrix}$$

Or in matrix notation: $\{u^e\} = [A]\{a\}$

$$\therefore \; u(x,y) = [x]\,[A]^{-1}\,\{u^e\} = [N]\,\{u^e\}; \qquad \therefore \; [N] = [x]\,[A]^{-1}$$

And $\quad u(x,y) = N_1\,u_1 + N_2\,u_2 + N_3\,u_3$

N_1, N_2 and N_3 are linear interpolation polynomials in x and y, which are given by:

$$N_i = \frac{1}{2\Delta}(a_i + b_i x + c_i y), \quad i = 1, 2, 3$$

where $a_i = x_j\, y_k - x_k\, yi_j$

$b_i = y_j - y_k$

$c_i = x_k - x_j$

and i,j and k are to be taken in a cyclic order. For example, when $i = 1$, j and k should be 2 and 3 respectively.

And

$$\Delta = \begin{vmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \end{vmatrix} = 2 \times \text{Area of the triangle}$$

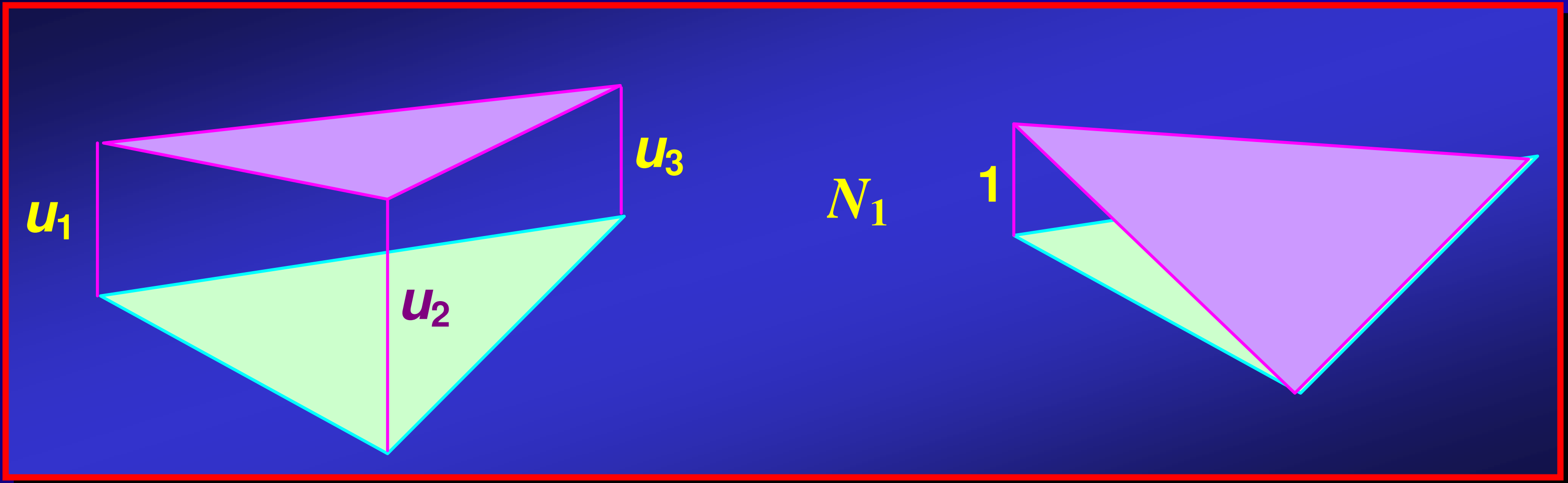


Figure: Interpolation polynomial N_1 for triangular element

Thus, we have seen now that if the nodal displacement vector is known, we can calculate the displacement at any point within any element.

That is, we can determine the displacement field once we know the nodal displacements.

The Strain-Displacement Relations

Once we know the displacement at any point, we can get the strain vector at the point by invoking the strain-displacement relations as given by:

$$\varepsilon_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)$$

For a two-dimensional elasticity problem, the components of the strain tensor are given by:

$$\varepsilon_x = \frac{\partial u}{\partial x}; \;\; \varepsilon_y = \frac{\partial v}{\partial y}; \;\; \gamma_{xy} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}$$

In matrix notation:

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \begin{Bmatrix} u \\ v \end{Bmatrix}$$

Or $\{\varepsilon\} = [L]\ \{u\}$

Since $\{u\} = [N]\ \{u^e\}$,

we have, $\{\varepsilon\} = [L]\ \{u\} = [L]\ [N]\ \{u^e\}$

Or $\{\varepsilon\} = [B]\ \{u^e\}$

where $[B]$ is called the strain-displacement matrix

The Constitutive Relations

The stress-strain relations are given in matrix form as follows:

$$\{\sigma\} = [D]\ \{\varepsilon\}$$

$[D]$ is called the constitutive matrix.

For plane stress problems:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \frac{E}{(1-\nu^2)} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1}{2}(1-\nu) \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix}$$

where E – Young’s modulus; ν – Poisson’s ratio

For the case of a three dimensional (linearly elastic isotropic) problem, the stress strain relations are given by:

$$\sigma_{ij} = \lambda \varepsilon_{kk} \delta_{ij} + 2\mu \varepsilon_{ij}$$

If there is an initial stress field $\sigma_o(x)$ and an initial strain field $\varepsilon_o(x)$, the corresponding constitutive relationship is:

$$\{\sigma\} = [D](\{\varepsilon\} - \{\varepsilon_o\}) + \{\sigma_o\}$$

To recapitulate, we have the following relations:

$\{u\} = [N]\ \{u^e\}$	– displacements within any element
$\{\varepsilon\} = [B]\ \{u^e\}$	– strain within any element
$\{\sigma\} = [D]\ [B]\ \{u^e\}$	– stress within any element

Now, let us address the earlier question –
How do we get the nodal displacements in the first place?

Principle of Virtual Work

Consider a virtual displacement field denoted by δu. The corresponding nodal displacement vector is δu^e and,

$$\{\delta u\} = [N]\ \{\delta u^e\}$$

The virtual strain field produced by the above displacement field is denoted by $\delta\varepsilon$ and is related to the virtual nodal displacement vector by:

$$\{\delta\varepsilon\} = [B]\ \{\delta u^e\}$$

The principle of virtual work states that when a body is in equilibrium under the action of certain external loads, the external virtual work done by these loads over the virtual displacement field is equal to the internal virtual work done by the stresses over the virtual strain field.

Mathematically, the principle of virtual work can be written as:

$$\int_V \{\delta\varepsilon\}^T\{\sigma\}dV = \int_V \{\delta u\}^T\{b\}dV + \int_S \{\delta u\}^T\{p\}dV$$

$$\Rightarrow \quad \{\delta u^e\}^T \int_V [B]^T[D][B]dV$$

$$= \{\delta u^e\}^T \int_V [N]^T\{b\}dV + \{\delta u^e\}^T \int_S [N]^T\{p\}dV$$

Or

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

Thus, $[K]\{U\} = \{R\}$

where

$$[K] = \sum_{e=1}^{nElems} [k^e]; \quad \{R\} = \sum_{e=1}^{nElems} \{r^e\}$$

$$[k^e] = \int_V [B]^T [D][B] dV;$$

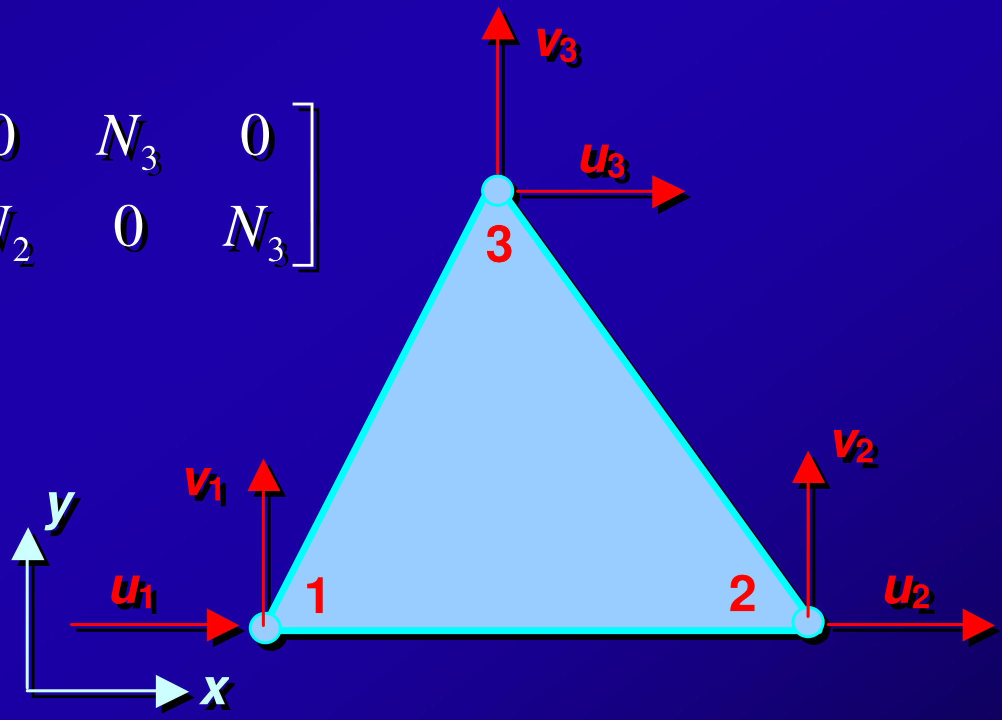
$$\{r^e\} = \int_V [N]^T \{b\} dV + \int_S [N]^T \{p\} dV$$

For the Triangular Element

The strain displacement matrix is given by,

$$[B] = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 \end{bmatrix}$$

$$= \begin{bmatrix} b_1 & 0 & b_2 & 0 & b_3 & 0 \\ 0 & c_1 & 0 & c_2 & 0 & c_3 \\ c_1 & b_1 & c_2 & b_2 & c_3 & b_3 \end{bmatrix}$$



It can be noticed that [*B*] is a matrix of constant elements.

Hence a constant strain field is obtained.

(hence, the name - *constant strain triangle*).

The element stress field is given by:

$$\{\sigma\} = [D]\,[B]\,\{u^e\}$$

where the elements of the 3×3 constitutive matrix $[D]$ are obtained from Eq. 2a or 2b depending on whether the problem is a plane stress one or a plane strain one.

The element stiffness matrix can be obtained as:

$$[k^e] = \int_{v_e} B^T DB \, dV_e = B^T DB \times \text{area of triangle} \times \text{thickness of element}$$

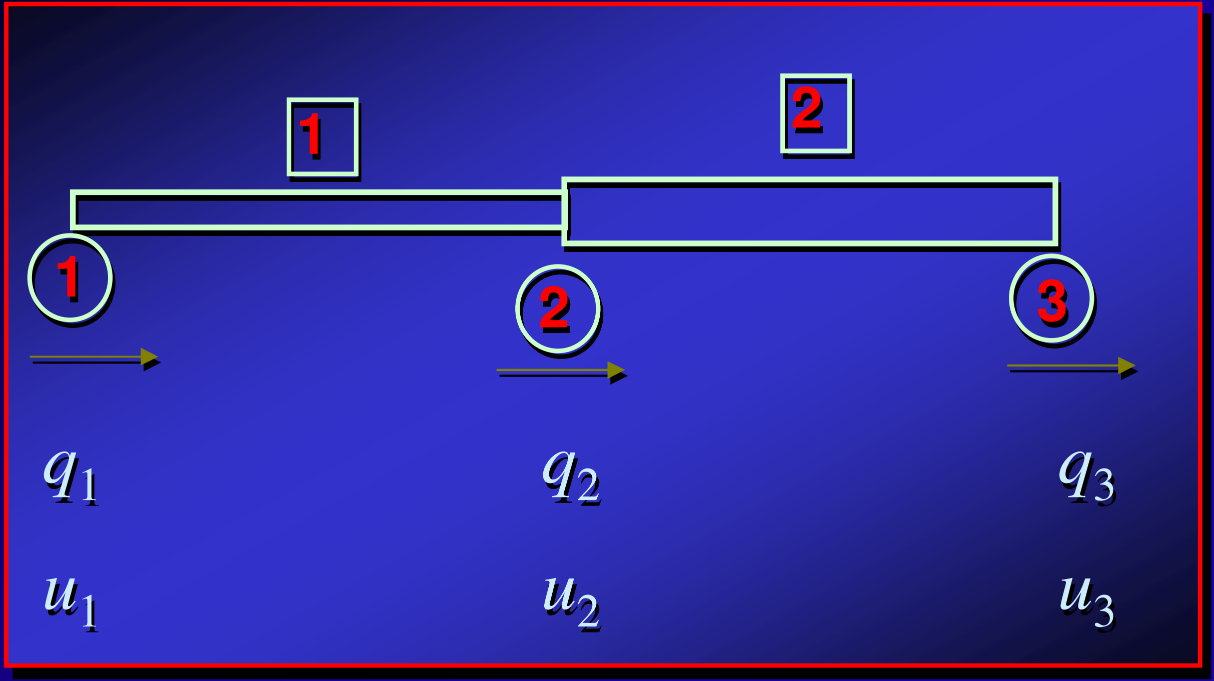
Explicit expression for k^e are available in standard text books.

Assembly of Element Matrices

Element stiffness matrix and load vector are calculated for each element and then assembled to get the global stiffness matrix and global load vector.

$$[K] = \sum_{e=1}^{nElems} [k^e]; \quad \{R\} = \sum_{e=1}^{nElems} \{r^e\}$$

Let us consider an example:



Nodal forces

Nodal displacements

$$\begin{Bmatrix} q_1^1 \\ q_2^1 \end{Bmatrix} = \begin{bmatrix} k_{11}^1 & k_{12}^1 \\ k_{21}^1 & k_{22}^1 \end{bmatrix} \begin{Bmatrix} u_1^1 \\ u_2^1 \end{Bmatrix}; \qquad \begin{Bmatrix} q_1^2 \\ q_2^2 \end{Bmatrix} = \begin{bmatrix} k_{11}^2 & k_{12}^2 \\ k_{21}^2 & k_{22}^2 \end{bmatrix} \begin{Bmatrix} u_1^2 \\ u_2^2 \end{Bmatrix}$$

Compatibility: $u_1{}^1 = u_1$; $u_2{}^1 = u_1{}^2 = u_2$; $u_2{}^2 = u_3$

Equilibrium: $q_1{}^1 = q_1$; $q_2{}^1 + q_1{}^2 = q_2$; $q_2{}^2 = q_3$

$$\begin{Bmatrix} q_1 \\ q_2 \\ q_3 \end{Bmatrix} = \begin{bmatrix} k_{11}^{1} & k_{12}^{1} & 0 \\ k_{21}^{1} & k_{22}^{1} + k_{11}^{2} & k_{12}^{2} \\ 0 & k_{12}^{2} & k_{22}^{2} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix}$$

This is of the form:

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

$[K]$ is singular. Suppose $u_3 = 0$.

Discarding the third row and column of K, we get:

$$\begin{bmatrix} K_{11} & K_{12} \\ K_{21} & K_{22} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} q_1 \\ q_2 \end{Bmatrix}$$

Find $\{u_1 \ u_2\}^{\mathrm{T}}$

Concept of Reduced Stiffness Matrix

The stiffness matrix, and displacement and load vectors can be partitioned as follows:

$$\begin{bmatrix} K_{ff} & K_{fr} \\ K_{rf} & K_{rr} \end{bmatrix} \begin{Bmatrix} U_f \\ U_r \end{Bmatrix} = \begin{Bmatrix} R_f \\ R_r \end{Bmatrix}$$

The first equation reads as:

$$\mathbf{K}_{ff}\,\mathbf{U}_f + \mathbf{K}_{fr}\,\mathbf{U}_r = \mathbf{R}_f$$

Most often, $\mathbf{U}_r = \mathbf{0}$. (We will consider the case of non-zero $\mathbf{U}_r$ later).

Then, $\mathbf{K}_{ff}\,\mathbf{U}_f = \mathbf{R}_f.$

$\mathbf{K}_{ff}$ is called the reduced stiffness matrix

Only the reduced stiffness matrix is assembled

The set of linear algebraic equations $[K_{ff}]\ \{U_f\} = \{R_f\}$ is solved to get $\{U_f\}$

Once $\mathbf{U}_f$ is known, the nodal displacement vector $\mathbf{u}^{\mathrm{e}}$ for each element can be extracted; the displacement, strain and stress fields within the element are then calculated using the earlier relations:

$$\{u\} = [N]\ \{u^{\mathrm{e}}\}$$

$$\{\varepsilon\} = [B]\ \{u^{\mathrm{e}}\}$$

$$\{\sigma\} = [D]\ [B]\ \{u^{\mathrm{e}}\}$$

The reactions are determined from each of the element contributions as:

$$\{r^{\mathrm{e}}\} = [k^{\mathrm{e}}]\ \{u^{\mathrm{e}}\}$$

Recapitulation

Steps in FE Analysis:

Discretise the domain into finite elements

Each element is connected to neighbouring ones only at nodes

The nodal displacements are the basic unknowns of the problem

For each element determine the element stiffness matrix using the following relation:

$$[k^e] = \int_V [B]^T [D][B] dV$$

and the element load vector by:

$$\{r^e\} = \int_V [N]^T \{b\} dV + \int_S [N]^T \{p\} dV$$

Assemble the element stiffness and load matrices to get the global stiffness matrix and global load array as:

$$[K] = \sum_{e=1}^{nElems} [k^e]; \quad \{R\} = \sum_{e=1}^{nElems} \{r^e\}$$

In the above, one need assemble only the reduced stiffness matrix (corresponding to the free degrees of freedom alone) and the corresponding load vector to obtain the system of equations:

$$[K_{ff}]\{U_f\} = \{R_f\}$$

Solve the above using Gauss elimination to determine the unknown nodal displacements

Extract element nodal displacements

Calculate the displacements, strains and/or stresses within any element as required

Programming Aspects of Finite Element Method

Elements of a simple FE computer code are presented below – "pseudo code" is used

Input requirements:

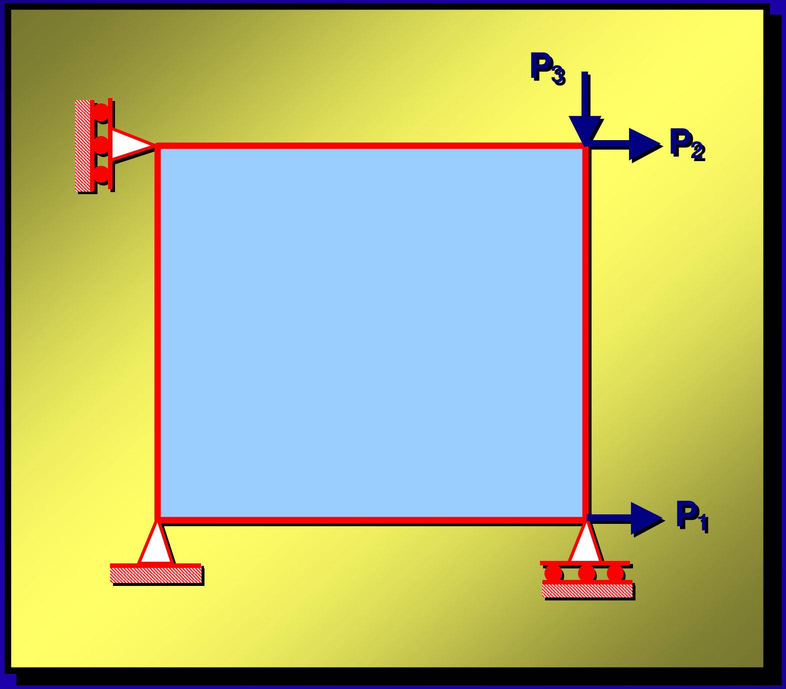
```
// Input no. of elements, no. of nodes, material properties
        read nElems, nNodes, E, poissonRatio
// Input nodal coordinate data
        for i=1, nNodes
        read k, x[k], y[k]
// Input element connectivity and thickness data
        for i=1, nElems
        read k, nod(1,k), nod(2,k), nod(3,k), thick[k]
```

For the purpose of assembly, the following two matrices are usually employed:

Element connectivity array: Array containing the node numbers of each element

Destination array: Array containing the degree-of-freedom numbers at each node

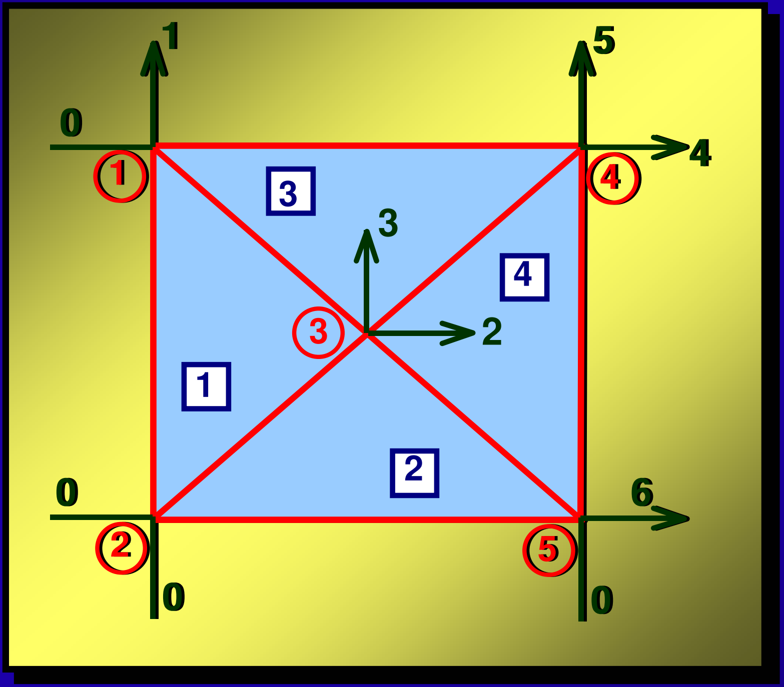
Consider the following plane stress problem:



Example –
Plane stress problem

Nodal connectivity array:

$$nod = \begin{bmatrix} 1 & 2 & 1 & 3 \\ 2 & 5 & 3 & 5 \\ 3 & 3 & 4 & 4 \end{bmatrix}$$



Destination array is initialized to zero; the boundary restraint list is then directly read into it to get:

$$destn = \begin{bmatrix} 1 & 1 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 1 \end{bmatrix}$$

It is modified subsequently to:

$$destn = \begin{bmatrix} 0 & 0 & 2 & 4 & 6 \\ 1 & 0 & 3 & 5 & 0 \end{bmatrix}$$

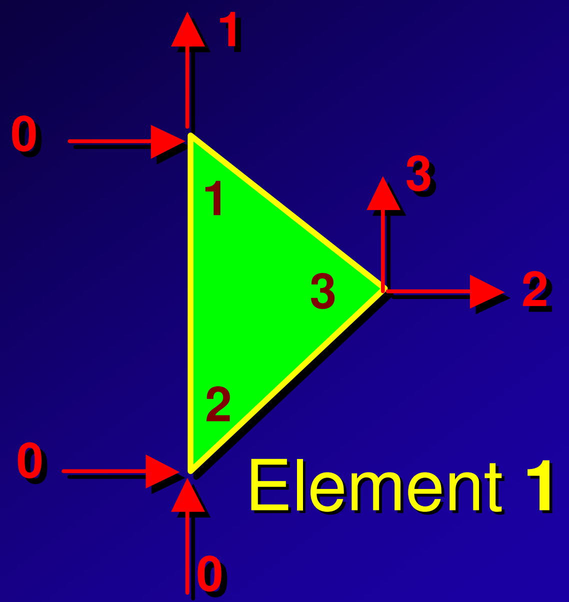
The global stiffness matrix will be of size 6 x 6

$$
[K] = \begin{array}{cccccc|c}
1 & 2 & 3 & 4 & 5 & 6 & \\
\hline
x & x & x & x & x & x & 1 \\
x & x & x & x & x & x & 2 \\
x & x & x & x & x & x & 3 \\
x & x & x & x & x & x & 4 \\
x & x & x & x & x & x & 5 \\
x & x & x & x & x & x & 6
\end{array}
\qquad
\{R\} = \left\{ \begin{array}{c} x \\ x \\ x \\ x \\ x \\ x \end{array} \right\}
\begin{array}{c} 1 \\ 2 \\ 3 \\ 4 \\ 5 \\ 6 \end{array}
$$

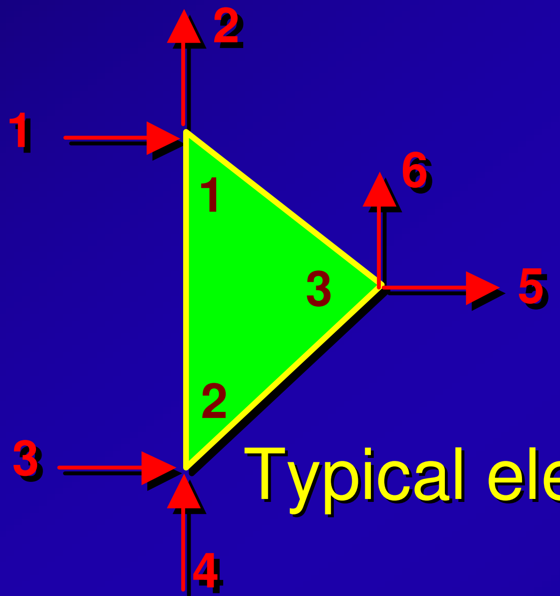
The element stiffness matrix is also of size 6 x 6

$$[k^e] = \begin{bmatrix} k_{11} & k_{12} & k_{13} & k_{14} & k_{15} & k_{16} \\ k_{21} & k_{22} & k_{23} & k_{24} & k_{25} & k_{26} \\ k_{31} & k_{32} & k_{33} & k_{43} & k_{35} & k_{36} \\ k_{41} & k_{42} & k_{43} & k_{44} & k_{45} & k_{46} \\ k_{51} & k_{52} & k_{53} & k_{54} & k_{55} & k_{56} \\ k_{61} & k_{62} & k_{63} & k_{64} & k_{65} & k_{66} \end{bmatrix}$$

We need to post the elements of the element stiffness matrix into the global stiffness matrix. The k_{ij}^e element gets posted to the location K_{pq} which is decided by the nodal d.o.f. numbering of each element as follows:



Element **1**



Typical element **e**

Local dof	1	2	3	4	5	6
Global dof Elem 1	0	1	0	0	2	3
Elem 2	0	0	6	0	2	3
Elem 3	0	1	2	3	4	5
Elem 4	2	3	6	0	4	5

(Input continued):

```
// input boundary restraint list
        read nDispRestrains
        for i=1, nDispRestrains
        read k, destn(1,k), destn(2,k)

// generation of destination array
        neq = 0
        for (j=1; j<=nNodes; ++j)
                for i=1, 2
                    if (destn (i, j) = 0)
                        neq ++;
                         destn (i, j) = neq;
                    else
                         destn (i, j) = 0;
                    end if
```

```
// Assembly of element matrices
    for  n = 1, nElems
        getElementStiffness(n);   // call subprogram
        dof = 0;
        for  l = 1, 3
          for  j = 1, 2
            dof = dof + 1
            node = nod(i, n)
        kk[dof] = destn (j, node)
        for i = 1, 6
          if (kk [i] > 0) then
            k = kk [i]
            gLoad [k] += elemLoad [i]
            for j=1, 6
              if (kk [j] >  0) then
                l = kk [j]
                gStiff  (k, l) += elemStiff  (i, j)
              end if
           end if
```

Prescribed Nonzero D.O.F.

We have, $K_{ff}\, U_f + K_{fr}\, U_r = R_f$

$$K_{ff}\, U_f = R_f - K_{fr}\, U_r$$

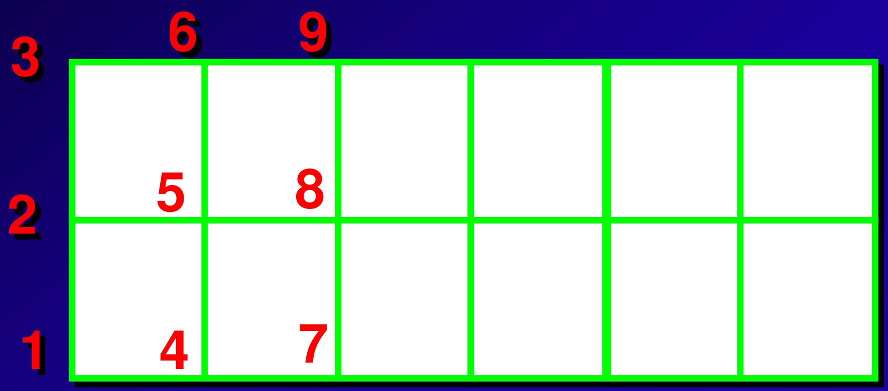
Specifically, as each element is assembled, calculate loads $\bar{r} = [k^e]\,\{d\}$ produced by prescribed d.o.f. $\{d\}$;

Subtract $\bar{r}$ from element loads $\{r^e\}$ and assemble the net loads

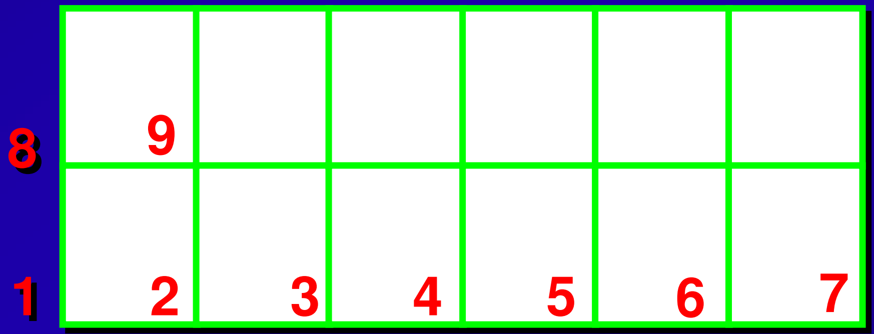
Sparsity of $[K]$ – Use of Proper Node Numbering

- $[K]$ is a *sparse* matrix (i.e., most of k_{ij} are zeros)
- The no. of nonzero elements of $[K]$ and their magnitudes are not affected by node numbering – only the arrangement of the nonzero k_{ij}’s is affected
- k_{ij} is nonzero if and only if the dof’s i and j are both present in at least one element
- Semi-bandwidth b (bandwidth is $2b - 1$)
- Some zeros might appear within the bandwidth also; but only zeros appear outside
- Sky-line storage method

To reduce the bandwidth *b*, number the nodes along the shorter dimension of the structure:



Good node numbering



Bad node numbering

- Automatic node renumbering programs

Stress Computation

Either use $\{\sigma\} = [D]\ \{\varepsilon\}$

or use: $\{\sigma\}=[D](\{\varepsilon\}-\{\varepsilon_o\})+\{\sigma_o\}$

Since $[B]$ involves differentiation, the stresses computed are less accurate in comparison with the displacement

In lower order elements, stresses are most accurate at the centroid of the element and least accurate at the corners

In higher order elements, there are points at which stresses are obtained with maximum accuracy (optimal points)

Stresses at other locations are obtained by extrapolation from these points

At nodes, it is better to calculate the stresses from various adjoining elements meeting at the node than from one element

For isoparametric elements, stresses are best calculated at the guassian points

Support Reactions

$$\begin{bmatrix} K_{ff} & K_{fr} \\ K_{rf} & K_{rr} \end{bmatrix} \begin{Bmatrix} U_f \\ U_r \end{Bmatrix} = \begin{Bmatrix} R_f \\ R_r \end{Bmatrix}$$

Either store K_{rf} and K_{rr} initially itself and use them to get R_r

A particular reaction R_i can be obtained as:

$$R_i = \sum_j K_{ij} U_j$$

$$= \sum_m \left(\sum_j K_{ij} U_j \right)$$

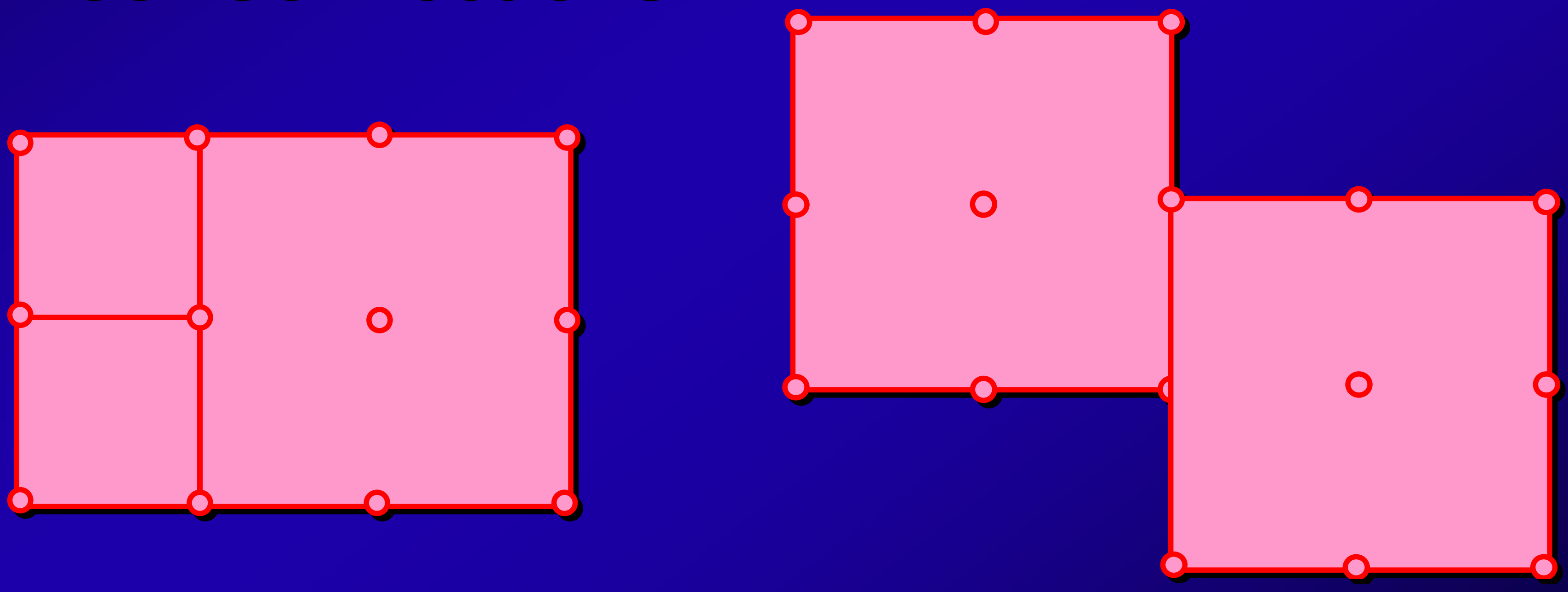
m – number of elements joined at the node

Element Shapes, Connection, Grading

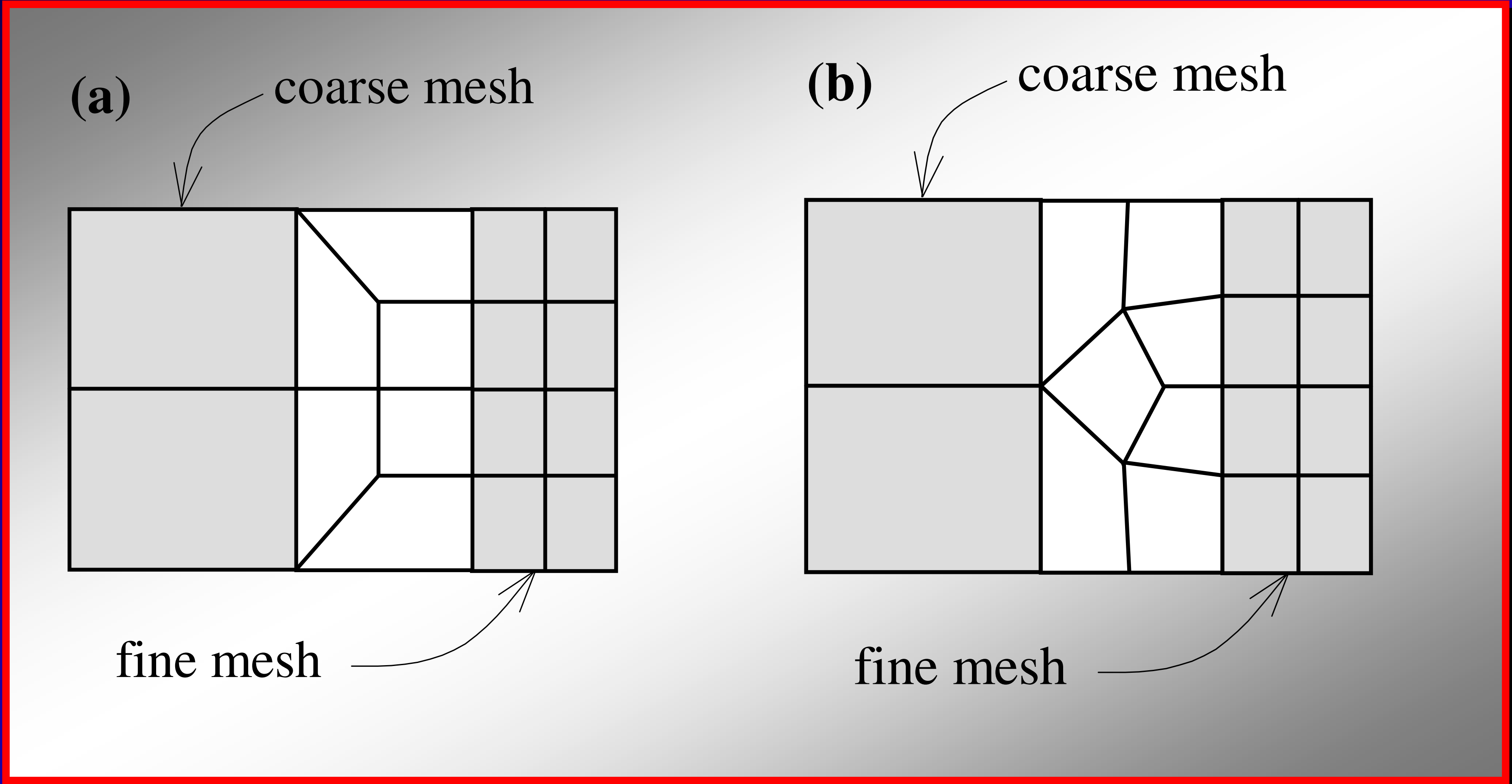
Best to use regular-shaped elements

Do not distort too much

Poor Connections



Transition from Coarse to Fine



Mesh for a Circular Domain