THE FINITE ELEMENT METHOD

7.1 GENERAL FORMULATION OF THE FINITE ELEMENT METHOD

In solving for the displacements, strains and stresses of any complex structure, we can use a *multi*-part Rayleigh-Ritz scheme by discretizing the structure under consideration into finite elements connected to each other at several nodes. Essentially what we are going to do is approximate the displacements inside each element by a “Rayleigh-Ritz function” with the “Rayleigh-Ritz parameters” being the nodal displacements. By calculating the energy stored in all these elements, and the virtual work associated with all applied loads, we can compute the potential energy functional for the complex structure in terms of the nodal displacements. By then minimizing the potential energy functional with respect to the nodal displacements, we get a system of equations that we can solve for the nodal displacements. The process is identical to what we have already discussed in one-dimension, but now we need to account for multi-dimensional displacements, strains and stresses.

· The elemental nodal displacement vector is:

$$\mathbf{d}_e^{\mathrm{T}} = \{u_1 \quad v_1 \quad w_1 \quad u_2 \quad v_2 \quad w_2 \quad \dots \quad \dots \quad \dots\}$$

where u_i,v_i and w_i represent 'x', 'y' and 'z' displacements respectively of node 'i' of element 'e', and contains as many slots as needed for each displacement direction at each node of the element (in two-dimensions you can drop the w's, and in one-dimension the v's as well; similarly in everything that follows). If there are ‘nnel’ nodes in an element and the degree of freedom (number of displacement components) at each node is ‘ndof’, there are edof=ndof*nnel element degrees of freedom, and so there are edof slots in $\mathbf{d}_\mathbf{e}$.

· The displacement function vector for this element is then given by:

$$\mathbf{u}_e(x,y,z) = \mathbf{N}_e \mathbf{d}_e$$

where $\mathbf{N}_\mathbf{e}$(x,y,z) is the shape function matrix and

$$\mathbf{u}_\mathrm{e}^{\mathrm{T}}(x,y,z) = \{u_x(x,y,z) \quad u_y(x,y,z) \quad u_z(x,y,z)\},$$

or whatever is appropriate to the dimension under consideration.

· The elemental strain vector with the necessary strain components (usually using engineering shear strains) can be expressed in terms of the nodal displacements as:

$$\varepsilon_\mathrm{e} = \mathbf{B}_e \mathbf{d}_e$$

where $\mathbf{B}_\mathbf{e}$ contains appropriate derivatives of the shape functions.

· The elemental stress vector is:

$$\sigma_e = \mathbf{D}_e \varepsilon_e$$

where $\mathbf{D_e}$ is the appropriate (1-d, 2-d plane stress, 2-d plane strain, or 3-d) elasticity matrix.

- The energy stored in the element can be written as:

$$U_e = \frac{1}{2}\int_{vol} \varepsilon_e^T \mathbf{D_e} \varepsilon_e dV = \frac{1}{2}\mathbf{d}_e^T \mathbf{K}_e \mathbf{d}_e$$

where

$$\mathbf{K}_e = \int_{vol} \mathbf{B}_e^T \mathbf{D}_e \mathbf{B}_e dV$$

is the element stiffness matrix.

- The total strain energy stored in all the elements is:

$$U = \sum_e \left\{ \frac{1}{2}\mathbf{d}_e^T \mathbf{K}_e \mathbf{d}_e \right\}$$
$$= \frac{1}{2}\mathbf{d}^T \mathbf{K} \mathbf{d}$$

where $\mathbf{K} = \sum_e \left\{ \mathbf{K}_e^{(aug)} \right\}$ is the **global stiffness matrix** obtained by summing the appropriately augmented elemental stiffness matrices, and

$$\mathbf{d}^T = \{u_1 \quad v_1 \quad w_1 \quad u_2 \quad v_2 \quad \ldots \quad \ldots \quad \ldots\}$$

is the **global nodal displacement vector** that contains (system degrees of freedom) sdof=ndof*nnode slots where 'nnode' is the total number of nodes in the entire discretized structure. Note that the augmentation of the elemental stiffness matrices must be to a square matrix of size sdof by sdof, and the elemental stiffness matrices are popped into the appropriate slots (determined by which nodes the element is attached to) of the augmented matrices.

- The potential energy functional can now be written as:

$$\Pi = \frac{1}{2}\mathbf{d}^T \mathbf{K} \mathbf{d} - \mathbf{d}^T \mathbf{F}$$

where $\mathbf{F}^T = \{F_{x1} \quad F_{y1} \quad F_{z1} \quad F_{x2} \quad F_{y2} \quad \ldots \quad \ldots \quad \ldots\}$ is the **global load vector** that gives the applied forces at each node along each direction. This is also of size sdof.

· Minimizing the potential energy functional with respect to the various nodal displacement components in **d** yields, as before: $\mathbf{Kd} = \mathbf{F}$.

· The finite element solution procedure can therefore be summarized as follows:

(i) Discretize the structure into nodes and elements, and choose the type of shape functions; thereby creating the global nodal displacement vector:

$$\mathbf{d}^{\mathrm{T}} = \{u_1 \quad v_1 \quad w_1 \quad \dots \quad \dots \quad \dots \quad u_{nnode} \quad v_{nnode} \quad w_{nnode}\}$$

which contains (system degrees of freedom, sdof)=(number of nodal displacement components) = (number of nodes,nnode)*(number of degrees of freedom at each node,ndof);

(ii) Calculate the elemental stiffness matrix and load vector for all the elements -- the integrations needed in this step are usually done numerically;

(iii) assemble the global unconstrained system stiffness matrix; that is: $\mathbf{K}$;

(iv) Impose constraints on nodal displacements and the applied loads to get the constrained system equations: $\mathbf{Kd} = \mathbf{F}$ -*constrained;*

(v) solve for the global nodal displacement vector using the constrained system equations;

(vi) from the global nodal displacement vector identify the elemental displacements at each node, and then use them to calculate the strains and stresses in the elements from $\varepsilon_e = \mathbf{B}_e \mathbf{d}_e$ and $\sigma_e = \mathbf{D}_e \varepsilon_e$.

This whole procedure can be readily implemented on a computer. We can use different kinds of elements to discretize a structure. Let us now look at some common elements.

7.2 THE LINEAR TRIANGULAR ELEMENT:

For plane problems (plane stress or plane strain), a two dimensional triangular element is often convenient. This is because it is rather easy to approximate any complicated planar cross-section by a bunch of connected triangles (see Fig. 1). This process of discretizing the actual geometry into elements and nodes is called "meshing."

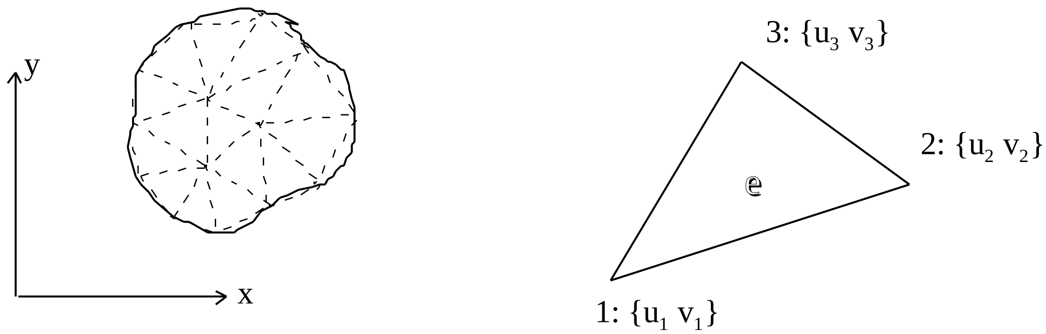


Figure 1

Consider an e'th triangular element consisting of three nodes (nodes per element: nnel=3). Let us call the nodes 1,2 and 3 - we can always map these to global node numbers through a *connectivity* matrix that tells us the global node numbers of every element. (That is, what we call the first, second and third nodes for an element above may actually be nodes 20, 32, 45 in a given finite element mesh etc.) For consistent bookkeeping purposes in the connectivity matrix we will always list the nodes in an element *counter-clockwise*. as shown in the figure. Let (x_i,y_i) i=1,2,3 be the nodal coordinates.

In two dimensions, there are two displacement components (ndof=2) at each of the three nodes of an element. Therefore there are edof=nnel*ndof=6 slots in the elemental nodal displacement vector. Next, we are going to assume that the displacements u_x and u_y inside the element vary linearly within the element. So, for the x-displacement:

$$u_x(x,y) = a_1 + a_2 x + a_3 y$$

Casting these in terms of the nodal displacements:

$$\begin{aligned} u_x(x_1,y_1) &= a_1 + a_2 x_1 + a_3 y_1 = u_1 \\ u_x(x_2,y_2) &= a_1 + a_2 x_2 + a_3 y_2 = u_2 \\ u_x(x_3,y_3) &= a_1 + a_2 x_2 + a_3 y_3 = u_3 \end{aligned} \qquad \text{or equivalently} \qquad \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \underbrace{\begin{bmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \end{bmatrix}}_{H} \begin{Bmatrix} a_1 \\ a_2 \\ a_3 \end{Bmatrix}$$

We can express the constants 'a' in terms of the nodal displacements and nodal coordinates, by inverting the above to get:

$$u_x(x,y) = \sum_{i=1}^{3} N_i(x,y)\ u_i \quad = N_1(x,y)\ u_1 + N_2(x,y)\ u_2 + N_3(x,y)\ u_3$$

$$N_1 = \frac{1}{2A}\left[\left(x_2y_3 - x_3y_2\right) + (y_2 - y_3)x + (x_3 - x_2)y\right]$$

where $N_2 = \frac{1}{2A}\left[\left(x_3y_1 - x_1y_3\right) + (y_3 - y_1)x + (x_1 - x_3)y\right]$

$$N_3 = \frac{1}{2A}\left[\left(x_1y_2 - x_2y_1\right) + (y_1 - y_2)x + (x_2 - x_1)y\right]$$

and $A = \frac{1}{2}\det\begin{bmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \end{bmatrix} = \frac{1}{2}\det[\mathbf{H}]$ is actually the area of the triangular element.

- Thus, using the *same* form for both the x- and y-displacements,

$$u_x(x,y) = a_1 + a_2x + a_3y \quad \text{and} \quad u_y(x,y) = b_1 + b_2x + b_3y$$

and determining the b's in terms of the nodal y-displacements v_1, v_2, v_3 as above, we have in general:

$$u_x(x,y,z) = \sum_{i=1}^{3} N_i(x,y,z)\ u_i \ ; \quad u_y(x,y,z) = \sum_{i=1}^{3} N_i(x,y,z)\ v_i$$

or equivalently stacking them up in matrix form:

$$\begin{Bmatrix} u_x \\ u_y \end{Bmatrix} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{Bmatrix}$$

or more concisely: $\mathbf{u}_e = \mathbf{N}_e\mathbf{d}_e$ where $\mathbf{N}_e$ is the shape matrix for a linear triangular element.

- The strain vector in two dimensions includes:

$$\begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \gamma_{xy} \end{Bmatrix} = \begin{Bmatrix} \frac{\partial u_x}{\partial x} \\ \frac{\partial u_y}{\partial y} \\ \frac{\partial u_x}{\partial y} + \frac{\partial u_y}{\partial x} \end{Bmatrix} = \begin{bmatrix} \frac{\partial N_1}{\partial x} & 0 & \frac{\partial N_2}{\partial x} & 0 & \frac{\partial N_3}{\partial x} & 0 \\ 0 & \frac{\partial N_1}{\partial y} & 0 & \frac{\partial N_2}{\partial y} & 0 & \frac{\partial N_3}{\partial y} \\ \frac{\partial N_1}{\partial y} & \frac{\partial N_1}{\partial x} & \frac{\partial N_2}{\partial y} & \frac{\partial N_2}{\partial x} & \frac{\partial N_3}{\partial y} & \frac{\partial N_3}{\partial x} \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{Bmatrix}$$

or more concisely: $\varepsilon_e = \mathbf{B}_e\mathbf{d}_e$, where:

$$\mathbf{B_e} = \begin{bmatrix} \frac{\partial N_1}{\partial x} & 0 & \frac{\partial N_2}{\partial x} & 0 & \frac{\partial N_3}{\partial x} & 0 \\ 0 & \frac{\partial N_1}{\partial y} & 0 & \frac{\partial N_2}{\partial y} & 0 & \frac{\partial N_3}{\partial y} \\ \frac{\partial N_1}{\partial y} & \frac{\partial N_1}{\partial x} & \frac{\partial N_2}{\partial y} & \frac{\partial N_2}{\partial x} & \frac{\partial N_3}{\partial y} & \frac{\partial N_3}{\partial x} \end{bmatrix}$$

which for the linear triangular element evaluates to:

$$\mathbf{B_e} = \left(\frac{1}{2A}\right) \begin{bmatrix} y_2 - y_3 & 0 & y_3 - y_1 & 0 & y_1 - y_2 & 0 \\ 0 & x_3 - x_2 & 0 & x_1 - x_3 & 0 & x_2 - x_1 \\ x_3 - x_2 & y_2 - y_3 & x_1 - x_3 & y_3 - y_1 & x_2 - x_1 & y_1 - y_2 \end{bmatrix}$$

Note that the above $\mathbf{B_e}$-matrix is a *constant* matrix for the linear triangular element. This means that the strains (and hence the stresses) are constant inside a linear triangular element. This is a direct consequence of our assumed linear shape function inside the triangular element. If, in a real problem, we expect that the strains and stresses are going to vary rather drastically in a region, what do you think we should do if we are restricted to using linear triangular elements?

- The elemental stiffness matrix is then easily computed from:

$$\mathbf{K_e} = \left\{ \frac{1}{2} \int_V \mathbf{B}_e^T \mathbf{D}_e \mathbf{B}_e \; dV \right\} = \mathbf{B}_e^T \mathbf{D}_e \mathbf{B}_e \, Ab$$

where we recognize that for a linear triangular element $\mathbf{B_e}$ and $\mathbf{D_e}$ are constant, and so the integral is just the volume, which is area $'A'$ times plate thickness $'b'$ (remember we are dealing with plane problems and nothing varies in the thickness-direction).Note that for other kinds of elements $\mathbf{B_e}$ may not be a constant through an element (ie the strains and stresses may vary through the element with such elements), and in such cases, the integration to determine the elemental stiffness matrix is best done numerically in a computer.

- The elasticity matrix $\mathbf{D_e}$ is given by:

for plane strain: $$\mathbf{D}_e = \frac{E(1-\nu)}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1 & \nu/(1-\nu) & 0 \\ \nu/(1-\nu) & 1 & 0 \\ 0 & 0 & (1-2\nu)/2(1-\nu) \end{bmatrix}$$

for plane stress: $$\mathbf{D}_e = \frac{E}{1-\nu^2}\begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & (1-\nu)/2 \end{bmatrix}$$

where E,ν are the Young's modulus and Poisson's ratio of the material. (Note that for an inhomogeneous body made of different materials, these can vary from element to element.)

Having computed the elemental stiffness matrix for every element, we can right away augment it and put it into the appropriate slots of the global stiffness matrix.

<u>Note</u>: The algebraic manipulations needed to get the shape function N-matrix and the kinematic B-matrix for a given element can be tedious. I have a MATLAB code **trielement.m** that does the algebra using the Symbolic Toolbox of MATLAB. You can easily modify it to do other elements such as the 4-node linear quadrilateral element for two dimensional problems, or 8-node linear brick elements for three-dimensional structures.

7.3 FINITE ELEMENT PACKAGES: There are several different kinds of elements that people have invented, and these are all coded in commercial FEM packages such as ANSYS, ABAQUS, NASTRAN etc. So, in practice, you will not really have to do all the algebra above if you are going to be just a *user* of finite element methods. What you will need to do is really just:

(i) create a finite element mesh for a structure that you want to analyze, ie:

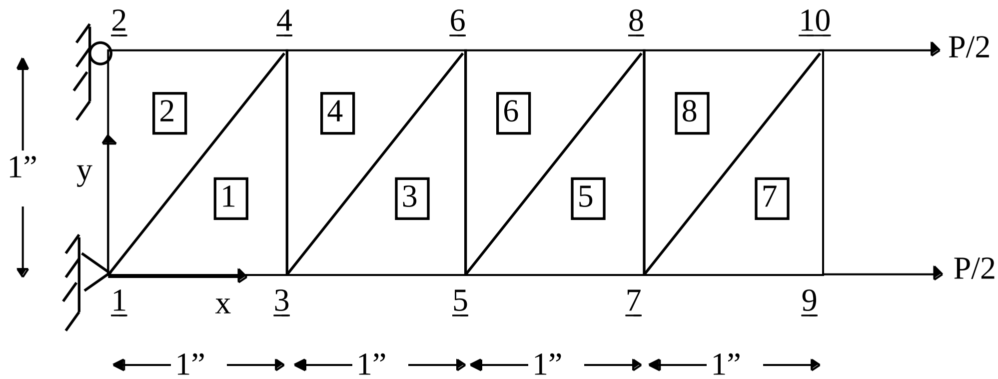
input the coordinates of the nodes of your structure;
tell the computer what nodes each element is attached to;
input the material properties (E,ν) for each element;
indicate whether you want a plane stress/strain, axisymmetric or 3-d analysis;
prescribe appropriate nodal displacement constraints;
prescribe the applied loads at the various nodes;

(ii) wait while the computer frets a bit calculating the global stiffness matrix and solving for the nodal displacements and elemental stresses and strains;

(iii) analyze the output results to make sure that the stresses and displacements do not exceed allowable/desired values.

EX1: AXIAL STRETCHING OF A ROD - PLANE STRESS ANALYSIS

(MESH: 8 triangular elements)

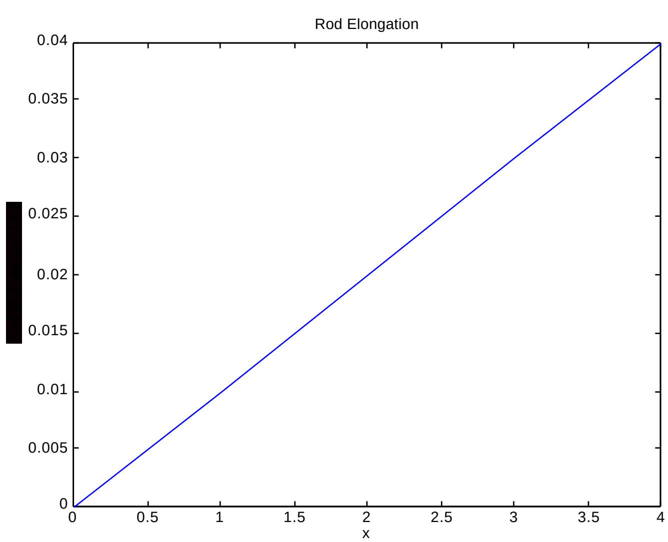


Input: $E=10^6$ psi, $\nu=0.3$; Thickness b=1". The applied normal stress on the free surface is 10^3 psi, which is equivalent to a net force of P=1000lbs, which is distributed equally between the nodes 9 and 10.

Output:

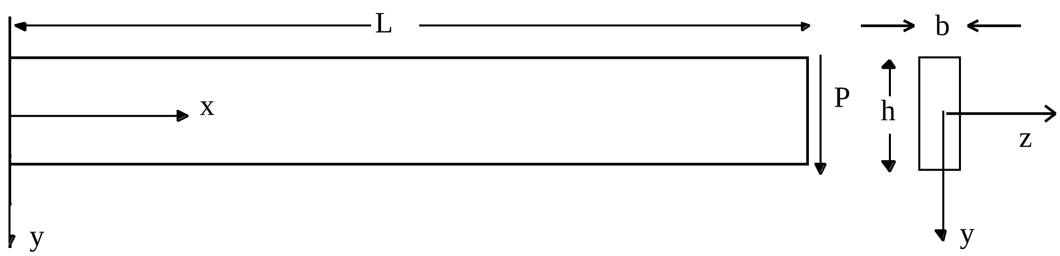
Stress: $\sigma_{xx}=1\text{x}10^3$psi; $\sigma_{yy}=0$; $\sigma_{xy}=0$;

Displacement: u_x[0 1 2 3 4"]=[0.0000 0.0100 0.0200 0.0300 0.0400"]



Remarks: Note that the results agree very well with the analytical solution.

EX2: CANTILEVER BEAM - PLANE STRESS ANALYSIS



Given: Material's Young's modulus is $E=1\times10^6$ psi, Poisson's ratio is $\nu=0.3$.

Geometry of beam: length L=4”; height h=1”; and thickness b= 1”.
Applied load: P=1000lb.

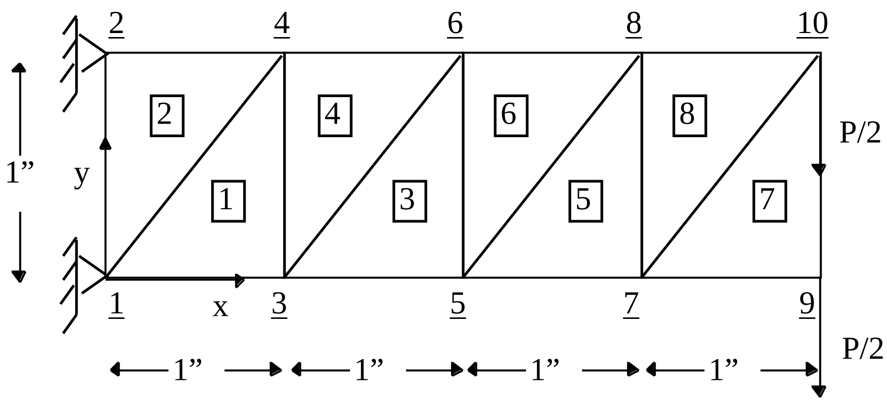
- The Bernoulli-Euler solution to the above cantilever beam is (do this on your own):

$$u_y(x) = \frac{PL^3}{2EI_z}\left\{\left(\frac{x}{L}\right)^2\right\}\left\{1-\frac{1}{3}\left(\frac{x}{L}\right)\right\}$$

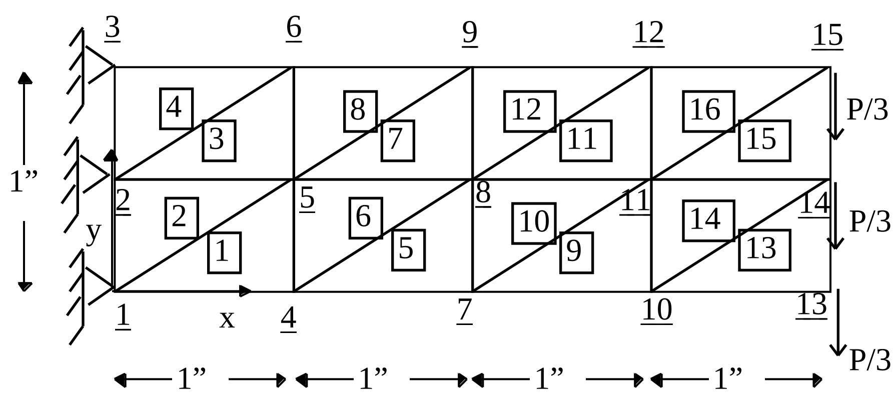
where the area moment of inertia for a rectangular cross-section is: $I_z = \dfrac{bh^3}{12}$.

- We can also solve the above problem as one of plane-stress using FEM. Let us consider different meshes. The last uses quadrilateral elements that you are looking into in a homework assignment.

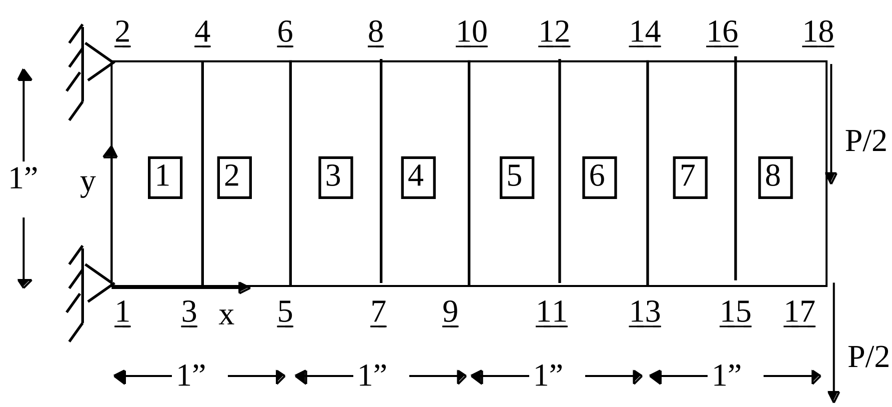
MESH 1: 8 Triangular elements



MESH 2: 16 Triangular elements



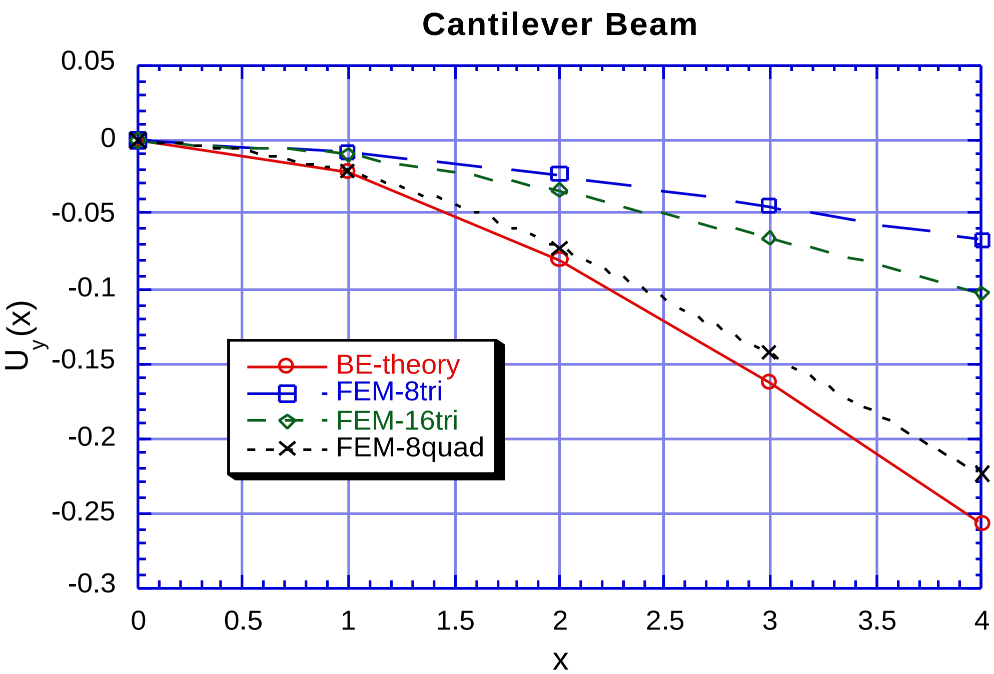
MESH 3: 8 Quadrilateral elements



<u>Output</u>: Beam Deflection

x	BE-theory*	FEM-8tri	FEM-16tri	FEM-8quad
0.0000	0.0000	0.0000	0.0000	0.0000
0.50000				-0.0060000
1.0000	-0.022020	-0.0077000	-0.0100000	-0.020800
1.5000				-0.043100
2.0000	-0.080160	-0.023300	-0.033500	-0.071720
2.5000				-0.10530
3.0000	-0.16254	-0.044300	-0.065800	-0.14270
3.5000				-0.18260
4.0000	-0.25728	-0.068100	-0.10240	-0.22380

* BE-theory results have been multiplied by -1 since the y-direction is switched between the usual BE coordinate frame and the ones used in plane-stress meshes.



Remarks:

(i) Why is the linear triangular element not good enough for beam bending, but did a good job in axial stretching? Reason: the stresses inside such elements are constant, but we know that the neutral axis in a beam is unstressed. So, mesh 1 should do a pretty bad job, and mesh 2 is not much better either.

(ii) The linear quadrilateral element allows for a linear variation in stress inside each element, and indeed from Bernoulli-Euler beam theory, that is precisely the variation of σ_{xx} along the y-direction. Therefore, only 8 quadrilateral elements do a much better job than 16 linear triangular elements.