Methods III:
Global Minimization Methods, rudiments of the Calculus of Variation, and Finite Element Methods

As outlined above, in the classical numerical treatment of PDEs -- the finite difference method -- the solution domain is approximated by a grid of nodes. At each node, the governing differential equation is approximated by an algebraic expression which references adjacent grid points. A system of equations is obtained by evaluating the previous algebraic approximations for each node in the domain. The system is finally solved for each value of the dependent variable at each node. In a sense, emphasis is placed on approximating the values of the exact solution at a *finite* number of mesh-points. In this section we concentrate on an alternative approach which is based on the *approximation of the exact solution by continuous piecewise functions*. In this Finite Element Method (FEM), the solution domain is also discretized. Importantly, the change of the dependent variable with regard to location is approximated *within* each element interconnecting nodes. This interpolation function is conveniently defined relative to the values of the variable at the nodes associated with each element. The original boundary value problem which is usually expressed in PDE-form is then replaced with an *equivalent integral formulation*, of a general type that we will introduce below. The interpolation functions in an element facilitate evaluation of the local integral equation. Once each element is integrated, the results from all other elements in the solution domain are combined. Conveniently, it turns out, the results of this procedure can be reformulated into a matrix equation, which is subsequently solved for the unknown variable at each node. Lastly, we note that the FEM was first proposed (by the physicist Courant) in 1943, but its importance was not recognised and the method was rediscovered by engineers in the early 1950's.

(modified from http://www.comlab.ox.ac.uk/internal/dow/endre/tmp/nspde/nspde.html and http://csep1.phy.ornl.gov/bf/node8.html)

Since *integral* minimization principles are central to the general application of the finite element interpolations, we will introduce these principles. The following topics are covered:

• Displacement solutions of two serial springs: Force analysis vs. Potential Minimization

• The Calculus of Variations and Energy Minimization applied to a Soap Film (Module II)

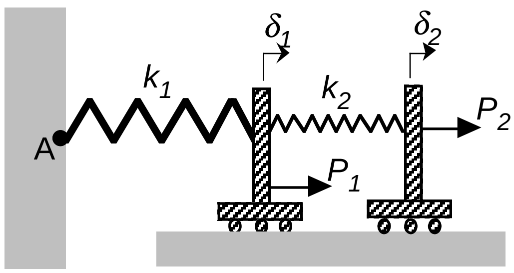
• Displacement solutions by FEM of a continuum solid analog to the simple, two-spring system

• FEM Basis Functions & an Integral Minimization Approach to Steady State Heat Conduction

For the simple, static system below, let's start with a determination, using force ideas, of the nodal displacements (δ_1 and δ_2) in terms of the applied loads (P_1 and P_2). The forces *in* springs 1 and 2 are seen by inspection to be, respectively:

$$F_1 = P_1 + P_2$$

$$F_2 = P_2$$



But we also readily see that

$$F_1 = k_1 \, \delta_1$$

$$F_2 = k_2 \, (\delta_2 - \delta_1).$$

Eliminating F_1 and F_2 from these four equations yields a solution

$$\delta_1 = (P_1 + P_2) \,/\, k_1 \qquad \text{Eq. 1a}$$

$$\delta_2 = P_2 \,/\, k_2 + \delta_1 \qquad \text{Eq. 1b}$$

Now let's use a very useful,"work - energy" approach in which we first define a potential, V, that is an "integrated" difference between the *strain energy stored* and the *work done* on the system:

$$V = \left(\tfrac{1}{2}\, k_1\delta_1^{\,2} + \tfrac{1}{2}\, k_2(\delta_2 - \delta_1)^2\right) \; - \; P_1\,\delta_1 \; - P_2\,\delta_2 \qquad \text{Eq. 2}$$

The first term is the strain energy built up in the springs under the work done by the external forces. The last two terms give this work done, noting that, though there is a net force of (P_1 + P_2) at the anchoring point A, there is no displacement at this constrained end and therefore no work done. If we then **minimize this potential** V with respect to each nodal displacement, we get

$$\partial V / \partial \delta_1 = 0 \quad \Rightarrow \quad k_1\delta_1 - k_2(\delta_2 - \delta_1) \; - \; P_1 = 0$$

$$\partial V / \partial \delta_2 = 0 \quad \Rightarrow \quad k_2(\delta_2 - \delta_1) \; - \; P_2 = 0$$

These two equations, by either subtraction or rearrangement, give exactly the same load-displacement relations as obtained above (Eqs. 1) by the force-based analyses :

$$\delta_1 = (P_1 + P_2) / k_1 \qquad \text{Eq. 3a}$$

$$\delta_2 = P_2 / k_2 + \delta_1 \qquad \text{Eq. 3b}$$

For use in continuum mechanics, this simple example must be "generalized" to loadings, displacements, and structures that are more continuous fields. This requires the “calculus of variations".

PROBLEM. To the right of the second node in the linear system above, add a third node that is displaced by δ_3 with externally applied load P_3. Determine the analogous load-displacement equations to Eqs.3.

Calculus of Variations

The *basic* problem is to find the y(x) that minimizes (or extremizes) an integral of a function, $f = f\{y, y'; x\}$, over some domain. The value of the integral, for whatever f, will be denoted as E:

$$E = \int_{x_1}^{x_2} f\{y, y'; x\}\, dx$$

Remember that E is a number, a scalar which we seek to minimize. It could be an energy if f is a strain energy density and y(x) is a spatially-dependent strain. Technically, f is a function of a function and is therefore given the special name "functional".

The strategy for minimizing E is to consider the most minimizing y(x) as being within the following family of "trial functions"

$$y(\alpha, x) = y(\alpha{=}0, x) + \alpha\eta(x)$$

In fact, this most minimizing y(x) is defined as $y(\alpha{=}0, x)$, and α is a "variational parameter", a parameter which we vary for all functions $\eta(x)$ that satisfy $\eta(x_1) = \eta(x_2) = 0$. The problem can be restated as

$$\text{for all } \eta(x), \qquad \partial E / \partial\alpha = 0 \quad \text{at} \quad \alpha = 0$$

Since α and x are independent variables, we can pull the differentiation inside the integral

$$\partial E / \partial\alpha = \int_{x_1}^{x_2} [\, (\partial f / \partial y)\, (\partial y / \partial\alpha) + (\partial f / \partial y')\, (\partial y' / \partial\alpha) \,]\, dx$$

Noting that $(\partial y / \partial\alpha) = \eta(x)$ and that $(\partial y' / \partial\alpha) = \eta'(x)$, we integrate the second term by parts. [Recall that $\int_a^b u\, dv = [uv|_a^b - \int_a^b v\, du\,]$. Noting that $\eta|_{\eta_1}^{\eta_2} = 0$, this gives

$$\begin{aligned}
\partial E / \partial\alpha &= \int_{x_1}^{x_2} (\partial f / \partial y)\, \eta\, dx + \int_{\eta_1}^{\eta_2} (\partial f / \partial y')\, d\eta \\
&= \int_{x_1}^{x_2} (\partial f / \partial y)\, \eta\, dx + \quad [(\partial f / \partial y')\, \eta\, |_{\eta_1}^{\eta_2} \; - \int_{x_1}^{x_2} \eta \, \frac{d}{dx}(\partial f / \partial y')\, dx \\
&= \int_{x_1}^{x_2} [\, (\partial f / \partial y) - \frac{d}{dx}(\partial f / \partial y') \,]\, \eta\, dx
\end{aligned}$$

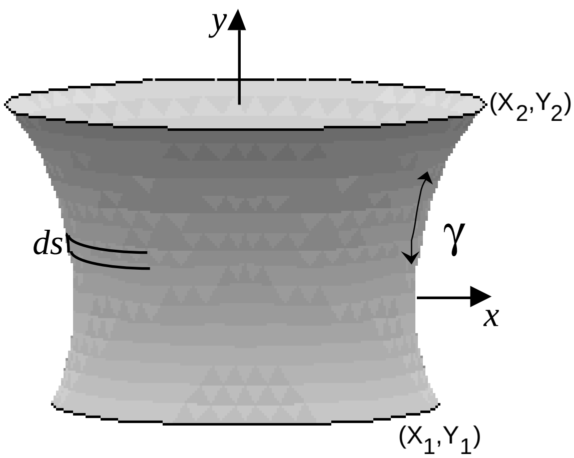
We now apply the minimization condition saying that the $\partial E / \partial\alpha = 0$ at $\alpha = 0$ for all $\eta(x)$. For this to be true for all $\eta(x)$, it's not difficult to see that the integrand must vanish, i.e.

$$(\partial f / \partial y) - \frac{d}{dx}(\partial f / \partial y') = 0. \qquad \text{(Euler's Eqn.)}$$

We've thus reduced our minimum integral problem to a differential equation.

To see how the above expressions work, we will now consider a problem with relatively simple mechanics. A soap film, like that between two rings below, is essentially a two-dimensional fluid in a three-dimensional space. The evidence for this is that if you blow a bubble and could put a drop of food color on it, you would see the food color diffuse rapidly in the film; more definitively, if you look very closely, you can sometimes see convection currents in such films. Analogous to a fluid in static equilibrium then, one expects the stress to be just a scalar pressure, and, since the film is thin, it makes the most sense to integrate across the film thickness and refer to a stress resultant known as the surface tension. For a stable bubble, the surface tension (γ: force/length) must be positive -- relevant discussions of this appear in Fung (pg. 235-238) with a central expression being a slightly generalized Law of Laplace. This law says that twice the product of surface tension and mean film curvature must balance or equal the pressure difference across the bubble (pressure inside being greater than pressure outside). The factor of two arises because there is both an inner and an outer interface, and, it turns out that the air (or whatever medium) on either side of the soap bubble dictates a characteristic *and constant* surface tension for each interface.

Because surface tension is a force acting along a length orthogonal to the force, it is equivalent to (force * length in direction of force) / (length orthogonal to the force * length in direction of force). The numerator of this expression is clearly a strain energy (or work done by intrinsic γ), and the denominator is the area that stores the strain energy. Surface tension is therefore *also* an intrinsic strain energy density with units of energy per area. Therefore, the total or "global" strain energy stored in a soap film is given by $E = \int_A \gamma \, dA$. The only external forces that might do some work are pressure differences across the wall, but, for an open surface such as the curved "tube" below, there can be no pressure difference across the wall.



$$ds^2 = dx^2 + dy^2$$

$$ds = [\,1 + (y')^2\,]^{1/2}\,dx$$

$$dA = 2\pi\,x\,ds = 2\pi\,x\,[\,1 + (y')^2\,]^{1/2}\,dx$$

For axisymmetric surfaces such as the one above, the work-energy approach thus contains only the total strain energy which, using geometry, can be written in terms of an integral over x:

$$E\{y(x)\} = 2\,\pi\,\gamma \int_{x_1}^{x_2} x\,(\,1 + y'^2\,)^{1/2}\,dx$$

where we see that $f\{y, y'; x\} = x\,(\,1 + y'^2\,)^{1/2}$. Taking derivatives of f then yields Euler's Eqn for this problem, $(\partial f / \partial y) - \frac{d}{dx}(\partial f / \partial y') = 0$:

$$\frac{d}{dx}\,[\,x\,y' / (\,1 + y'^2\,)^{1/2}\,] = 0.$$

Integrating this once gives: $x\,y' / (\,1 + y'^2\,)^{1/2} = A$ (A is constant)

Isolating the y' - term yields: $y' = A / (\,x^2 - A^2\,)^{1/2}$

Integrating once more shows: $y = b + A \cosh^{-1}(x/A)$ (B is constant)

Finally, inverting the cosh: $x = A \cosh[\,(\,y - B\,) / A\,]$

The constants A and B are determined from the boundary conditions $y(X_1) = Y_1$ and $y(X_2) = Y_2$. The surface is referred to as a "catenoid". If y=0 is at the waist, then B=0 and x=A is the waist radius.

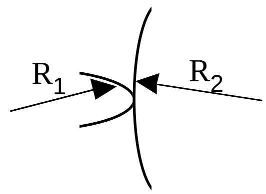
We have thus found the minimum energy (aka minimum area) soap film shape which continuously spans two parallel rings. In principle, however, one should next calculate the energy of the continuous film using the cosh function above and then compare this to a *discontinuous* soap film held separately as a disk within each ring: $E = \gamma\,(\pi\, r_1^2 + \pi\, r_2^2)$. In making such a comparison for a given pair of rings of equal radii, $r_1 = r_2$, that are separated by a distance h, one would actually find that h must be smaller than some critical height not much larger than r_1. You will be asked to demonstrate this two-disk to catenoid transition by numerically varying h in Module II.

One last comment to be made on the catenoid above is that it is, for perhaps obvious reasons, among a class of surfaces referred to as minimal surfaces. As a consequence (proofs are beyond the scope of this introduction) the *principal curvatures* of the film sum to zero everywhere on the surface. To understand this, recall that, for a plane curve, the circle which is tangent to the curve has a center located along the normal to the curve and the inverse of the radius of the circle is the curvature. However, on a curved *surface*, one can draw an infinite number of surface curves through a given point. The two *principal curvatures* are obtained though with those two (orthogonal) surface curves which yield the maximum and minimum radii of curvature, $1/R_1$ and $1/R_2$, respectively. In other words, for a minimal surface:

$$1/R_1 + 1/R_2 = 0$$

as Fung cursorily describes (pg. 237).

For the catenoid the easiest point to verify the formula above is a position at the waist where y = 0, the radius x=A is a minimum, and the surface normal is clearly in the x-y plane. This is illustrated below as the intersection of orthogonal surface curves. At this point, one curvature is just $1/R_1 = 1/A$. The second curvature is determined from the standard curvature expression in calculus texts for a plane curve, i.e. $1/R_2 = |y''| / (1 + y'^2)^{3/2}$, where $y = A \cosh^{-1}(x/A)$. Once evaluated at y=0 and x=A, one obtains $1/R_2 = 1/A$. Since the circles are on either side of the normal, they are actually of opposite sign so that $1/R_1 + 1/R_2 = 1/A - 1/A = 0$, as required.



Integral Principles of Finite Element Methods

Analogous to the energy minimization principle used to obtain the catenoid shape of a thin fluid film, the basic problem in solid mechanics can often be formulated as an attempt to find the displacement field $u_i(x, y, z)$ that minimizes (or extremizes) some work-energy potential integrated over an entire body under specified loads and/or boundary displacements. Essentially, one can think of displacing nodes, which simultaneously alter continuous interpolating fields between nodes, and then using these fields in the minimization. Other minimization principles are formulated in other types of problems in applied science and engineering.

In 1-D steady state heat conduction, for example, the relevant differential equation for the temperature $T(x; 0\leq x \leq 1)$, given a material conductance k and heat sink per volume $q(x, T)$, is

$$\frac{d}{dx}(-k\,T') + q = 0,$$

can be thought of as the Euler Eqn. in

$$\int_0^1 \left[\frac{d}{dx}(-k\,T') + q \right] \omega(x)\, dx = 0$$

where $\omega(x)$ is, in principle, an aribitrary function analogous to $\eta(x)$. If $T(x)$ were an exact solution to the original ODE, the integral would vanish because the term in square brackets vanishes. If, however, $T(x)$ is only an approximate solution, such as a set of linear interpolations between nodes, then only the integrated error can be thought of as vanishing, i.e. minimal, over the domain. There is skill required in choosing trial functions $T(x)$ and $\omega(x)$, as described for this heat conduction problem at the end of this Method III.

A common 2-D PDE in fluid and solid mechanics as well as diffusion and electrostatics involves a function $\phi(x,y)$, which may be a stress function (eg., in torsion of a bar, $\sigma_{xz} = \partial\phi/\partial y$ and $\sigma_{yz} = \partial\phi/\partial y$), a fluid stream function, an electrostatic potential, or even a concentration:

$$\nabla^2\phi = S(x,y),$$

This PDE can be considered the Euler Eqn. of

$$\int_0^1 [\, \nabla^2\phi - S \,]\, \omega(x)\, dx = 0.$$

Working backwards through the calculus of variations, this is found to come from minimizing

$$E\{\phi, \nabla\phi; x\} = \int_0^1 \left[\tfrac{1}{2}(\nabla\phi)^2 - S\,\phi \right] dx$$

In torsion, at least, the first term is readily seen to be proportional to the shear strain energy (Recall that $\sigma_{xz} = G\gamma_{xz}$). Finally, a more general integral used in 3-D linear, static elasticity of a solid V with surface A:

$$E\{u_i; x_i\} = \iiint_V \left[\tfrac{1}{2}(C_{ijkl}\, \varepsilon_{ij}\, \varepsilon_{kl})^2 - u_i b_i \right] dV - \iint_A [\, t_i\, u_i \,]\, dA$$

where t_i is a traction. An Euler eqn. that comes out of this is just the equation of motion,

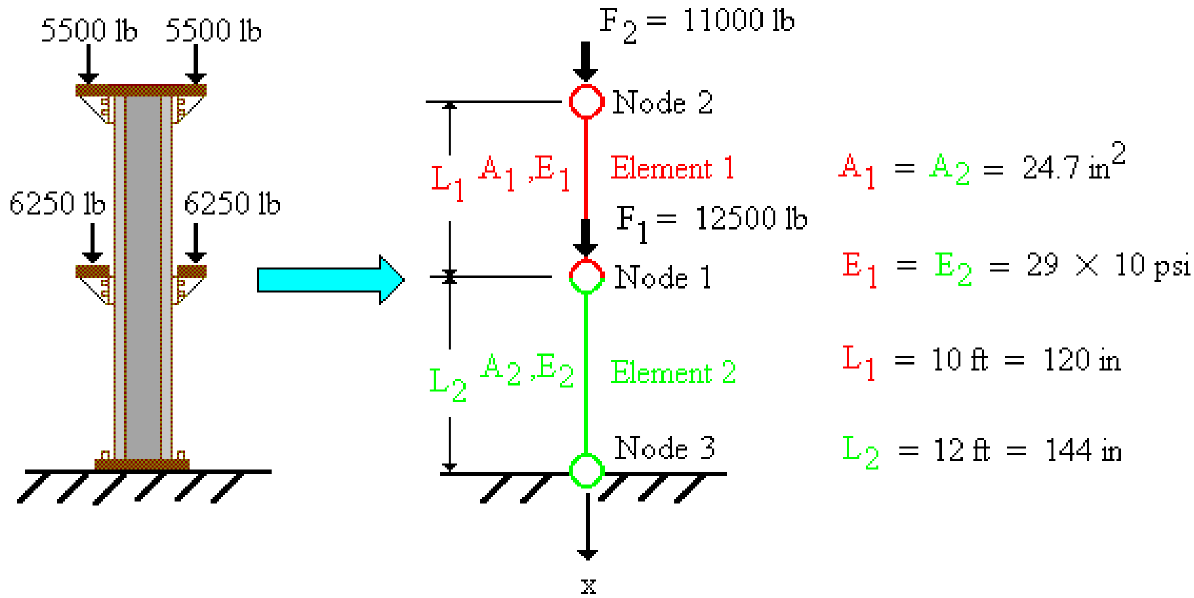
$$\sigma_{ij,j} + b_i = 0.$$

Example 1-D, Displacement Solution with a "Force-Based" FEM

from
http://femur.wpi.edu/Learning-Modules/Stress-Analysis/
One-Dimensional-Elements/Truss-Element/example.htm
by
JJ Rencis et al

The following problem is a continuum mechanics analog of the two spring system analysed earlier. It introduces the simplest finite element, a 1-D, linear truss, and illustrates the basic FEM concept of interpolating fields between model nodes as well as the construction and use of matrices in solutions. However, the basic method of solution employs only an analysis of forces rather than a minimization principle such as work-energy, although the latter will be seen to yield the same result.

Deformation of a Continuous Truss and its Finite Element Mesh:
A finite element mesh will be constructed for a 2-story truss in a building. Each side of the truss or column is subjected to a resultant concentrated load of (220 lb/ft)(50 ft)/2 = 5500 lb on the top floor and (250 lb/ft)(50 ft)/2 = 6250 lb on the middle floor. Since the loads are symmetric, they can be combined into concentrated forces as shown in the right-hand figure below.

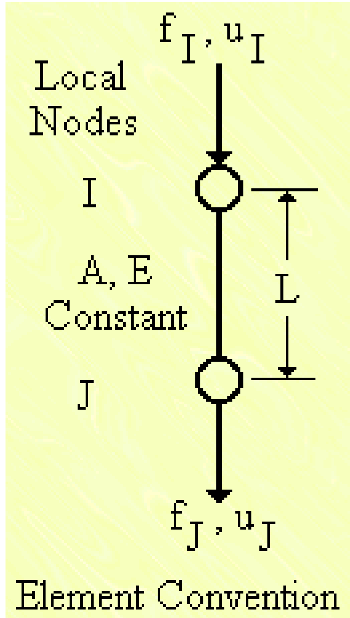


Actual Structure Finite Element Model

The simple finite element mesh consists of two elements and three nodes. The subscripts denote element quantities for all variables, except for the external forces F_1 and F_2 which denote nodal values. In general, additional one-dimensional bar elements must be introduced wherever there are changes in: Material Properties, Cross-Sectional Area, or, as in this case, External Loading. The external load applied at the midheight (12 feet from the ground) requires specifying two elements since all external loads in FEM must be applied at the nodal points. Note that one can assign the global nodes in any order to be convenient for the whole finite element mesh. In this case the global nodes are shown not in order to illustrate the flexibility.

Central to the FEM approach to modelling continuous materials, the displacements of material points along each truss element interpolate (linearly) and continuously between the nodes. This will clearly be seen in the final solution of displacement along the length.

1: Determine the stiffness matrix for each element.



The first thing that must be done is to determine the stiffness matrix for elements 1 and 2. Before this can be done we must obtain the stiffness matrix formulation for a one-dimensional bar element. The force-displacement relationship for one-dimensional bar element relates two nodal forces to two nodal displacements through a 2 x 2 element stiffness matrix. The element sign convention is that nodal forces and displacements are positive downward. In expanded form:

$$\underbrace{\begin{Bmatrix} f_I \\ f_J \end{Bmatrix}}_{\underline{f}_E} = \underbrace{\frac{AE}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}}_{\underline{K}_E} \underbrace{\begin{Bmatrix} u_I \\ u_J \end{Bmatrix}}_{\underline{u}_E}$$

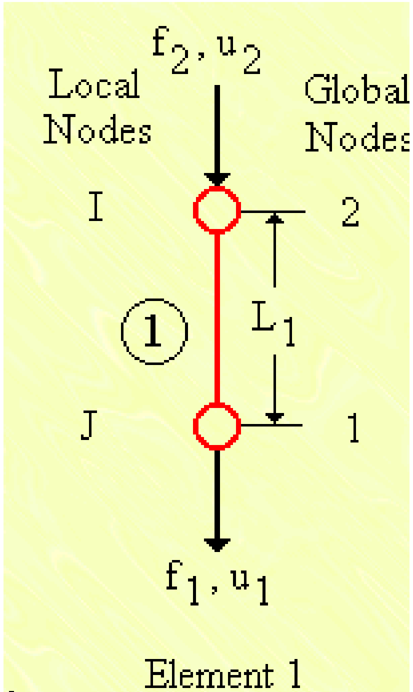
where

- f_I - Axial force of local node I.
- f_J - Axial force of local node J.
- u_I - Axial displacement of local node I.
- u_J - Axial displacement of local node J.
- A - Element cross-sectional area.
- E - Element modulus of elasticity.
- L - Element length.

In symbolic form the element force-displacement relationship is:

$$\underset{2\times 1}{\underline{f}_E} = \underset{2\times 2}{\underline{K}_E}\ \underset{2\times 1}{\underline{u}_E}$$

The numerical stiffness matrices for elements 1 and 2 will now be determined:



The local node numbers I and J must be mapped to the global node numbers in finite element mesh of the entire structure. The local and global node numbers are shown in the table and figure.

Element 1 Nodal Conductivity	
Local Nodes	**Global Nodes**
I	2
J	1

The geometric and material properties of a W14 x 18 steel column needed to find the stiffness matrix for element 1 are:

A = 24.7 in2
E = 29 x 10^6 psi
L = 120 in
AE/L = 5969166.67 lb/in

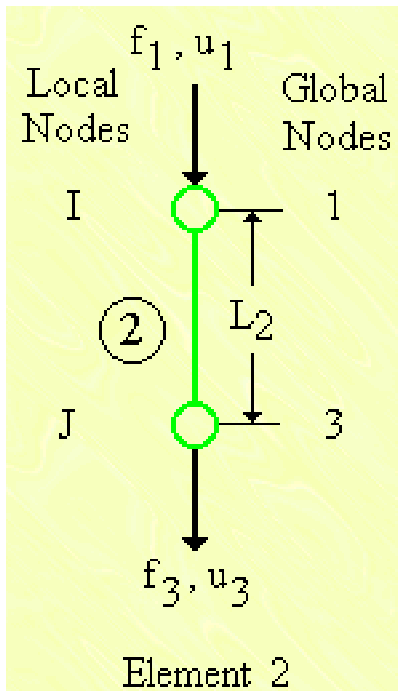
where consistent units have been used throughout.

The stiffness matrix of element 1 is therefore:

$$\underline{K}_1 = 5969166.67 \begin{array}{cc} \begin{array}{cc} 2 & 1 \end{array} & \\ \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} & \begin{array}{c} 2 \\ 1 \end{array} \end{array}$$

$$= \begin{array}{cc} \begin{array}{cc} 2 \qquad\qquad & \qquad 1 \end{array} & \\ \begin{bmatrix} 5969166.67 & -5969166.67 \\ -5969166.67 & 5969166.67 \end{bmatrix} & \begin{array}{c} 2 \\ 1 \end{array} \end{array}$$

where the numbers above the columns and to the right of the rows will be used for assembling the stiffness matrix of element 1 into the stiffness matrix of the entire structure in the next step.



Similar to element 1, the local node numbers I and J must be mapped to the global node numbers in finite element mesh of the entire structure. The local and global node numbers are shown in the table and figure.

Element 2 Nodal Conductivity	
Local Nodes	**Global Nodes**
I	1
J	3

The geometric and material properties of a W14 x 18 steel column needed to find the stiffness matrix for element 2 are:

A = 24.7 in2
E = 29 x 10^6 psi
L = 144 in
AE/L = 4974305.56 lb/in
where consistent units have been used throughout.

The stiffness matrix of element 2 is therefore:

$$\underline{\mathbf{K}}_2 = 4974305.56 \begin{array}{cc} \begin{array}{cc} 1 & 3 \end{array} & \\ \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} & \begin{array}{c} 1 \\ 3 \end{array} \end{array}$$

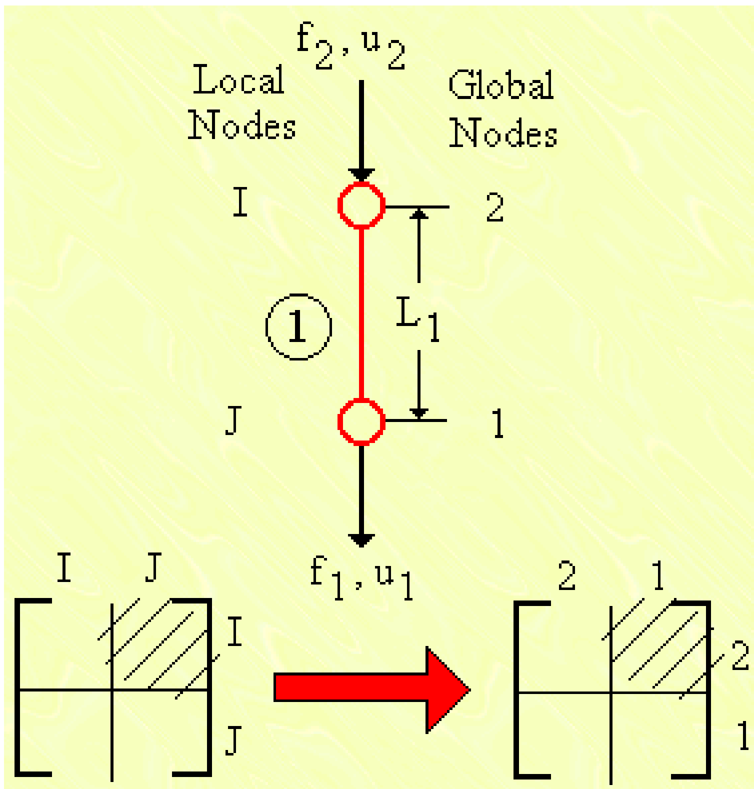
$$= \begin{array}{cc} \begin{array}{cc} 1 & 3 \end{array} & \\ \begin{bmatrix} 4974305.56 & -4974305.56 \\ -4974305.56 & 4974305.56 \end{bmatrix} & \begin{array}{c} 1 \\ 3 \end{array} \end{array}$$

2: Determine the stiffness matrix of the entire mesh.

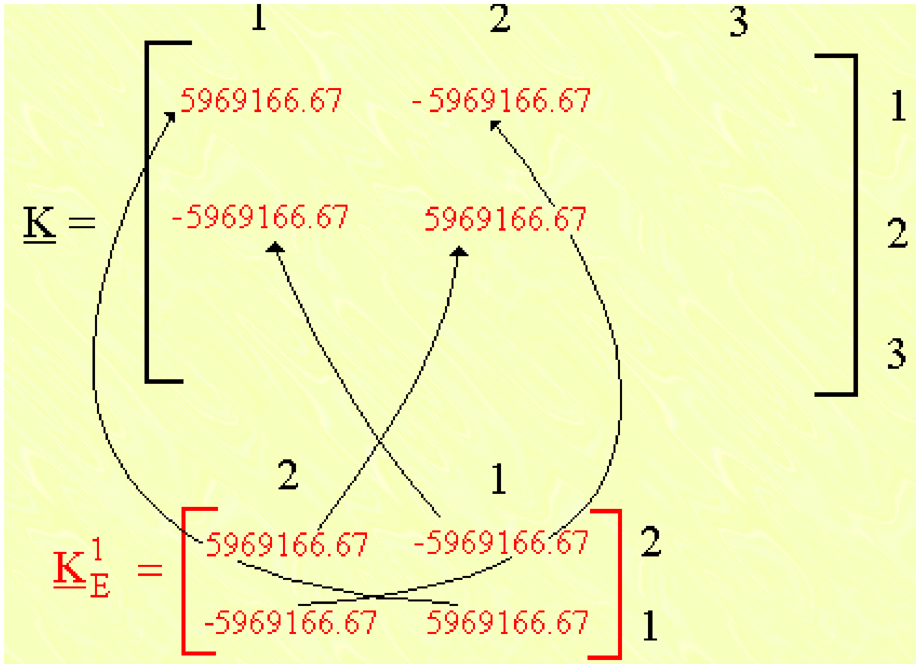
The present finite element mesh contains three nodes with one translational degree of freedom per node. The force-displacement relationship of the entire structure therefore relates three nodal forces to three nodal displacements through a 3 x 3 stiffness matrix. In symbolic form:

$$\underset{3\times1}{\underline{F}} = \underset{3\times3}{\underline{K}}\ \underset{3\times1}{\underline{u}}$$

where F is a column vector of external nodal forces and u is a column vector of nodal displacements. The element stiffness matrix is associated with the global node numbers as follows:

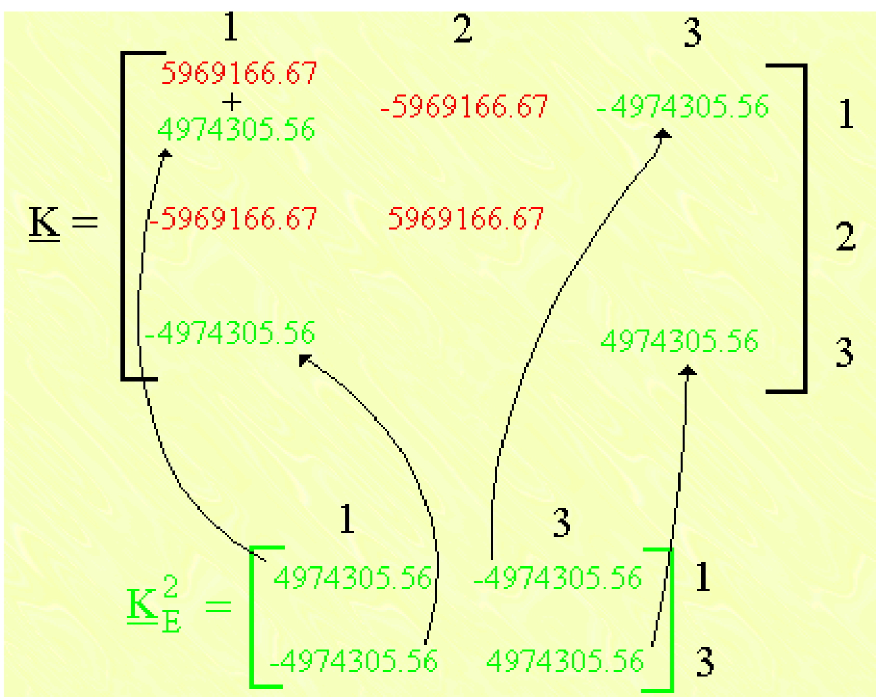


Thus the shaded "box" would be regarded as row 2-column 1 in the global stiffness matrix based on the element convention shown above. In other words, it relates the force at node 2 to the displacement at node 1. The stiffness matrices of elements 1 and 2 will be individually assembled into the stiffness matrix of the entire structure's K using a "direct assembly" procedure. First consider element 1 as follows:



For element 1, local nodes I and J are globally assigned as 2 and 1, respectively; thus to assemble the stiffness coefficients for element 1, the row and column numbers have to correspond with the row and column numbers in the global stiffness matrix. So the value in row 1-column 1 of will be inserted in the row 1-column 1 position in K. The same procedure is used to assemble row 1-column 2, row 2-column 1, and so on.

Note that the assembly procedure would be the same if the local node I and J of element 1 are assigned as global nodes 1 and 2, respectively. Now assemble element 2 as follows:



Again, the stiffness coefficient in row 1-column 1 of K_1^E will be inserted in the row 1-column 1 position of K. Since a value has been placed there from element 1, the inserted value will be added to the value from element 1. After the assembly procedure is completed, the empty positions in K are filled with zeros as follows:

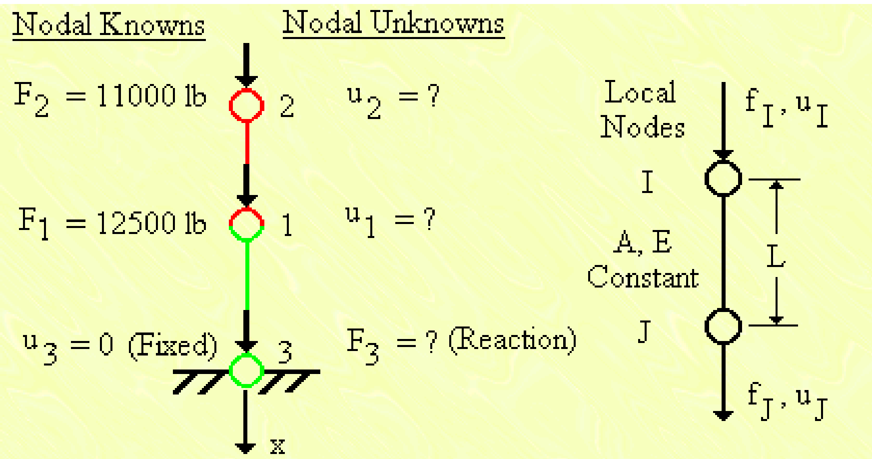
The zero term in K_{23} signifies that node 2 is not connected to node 3 (uncouple 3 from 2). The same applies to K_{32}

$$\underline{K} = \begin{array}{c} \begin{array}{ccc} 1 & 2 & 3 \end{array} \\ \begin{bmatrix} 10943472.23 & -5969166.67 & -4974305.56 \\ -5969166.67 & 5969166.67 & 0 \\ -4974305.56 & 0 & 4974305.56 \end{bmatrix} \end{array} \begin{array}{c} 1 \\ 2 \\ 3 \end{array}$$

4: Apply boundary conditions nodal-wise.

The boundary conditions are applied on a nodal basis. At a given node either the displacement is known or the external force is known, not both.

The boundary conditions are obtained from the mesh of the entire structure as shown in the figure on the left-hand side below. The sign convention for element nodal forces and displacements are based on the positive convention for the bar element shown below on the right-hand side.



Nodal Boundary Conditions Element Sign Convention

$$
\underset{3\times 1}{\begin{Bmatrix} F_1 = +12500 \\ F_2 = +11000 \\ F_3 = \ ? \end{Bmatrix}} = \underset{3\times 3}{\begin{array}{c} \begin{array}{ccc} 1 & 2 & 3 \end{array} \\ \begin{bmatrix} 10943472.23 & -5969166.67 & -4974305.56 \\ -5969166.67 & 5969166.67 & 0 \\ -4974305.56 & 0 & 4974305.56 \end{bmatrix} \end{array}} \begin{array}{c} 1 \\ 2 \\ 3 \end{array} \underset{3\times 1}{\begin{Bmatrix} u_1 = \ ? \\ u_2 = \ ? \\ u_3 = 0 \end{Bmatrix}}
$$

Since the solution of a system of linear algebraic equations requires all knowns be on the left-hand side and all unknowns on the right-hand side, i.e.

$$
\underset{n\times 1}{\underline{b}} = \underset{n\times n}{\underline{A}}\ \underset{n\times 1}{\underline{x}}
$$

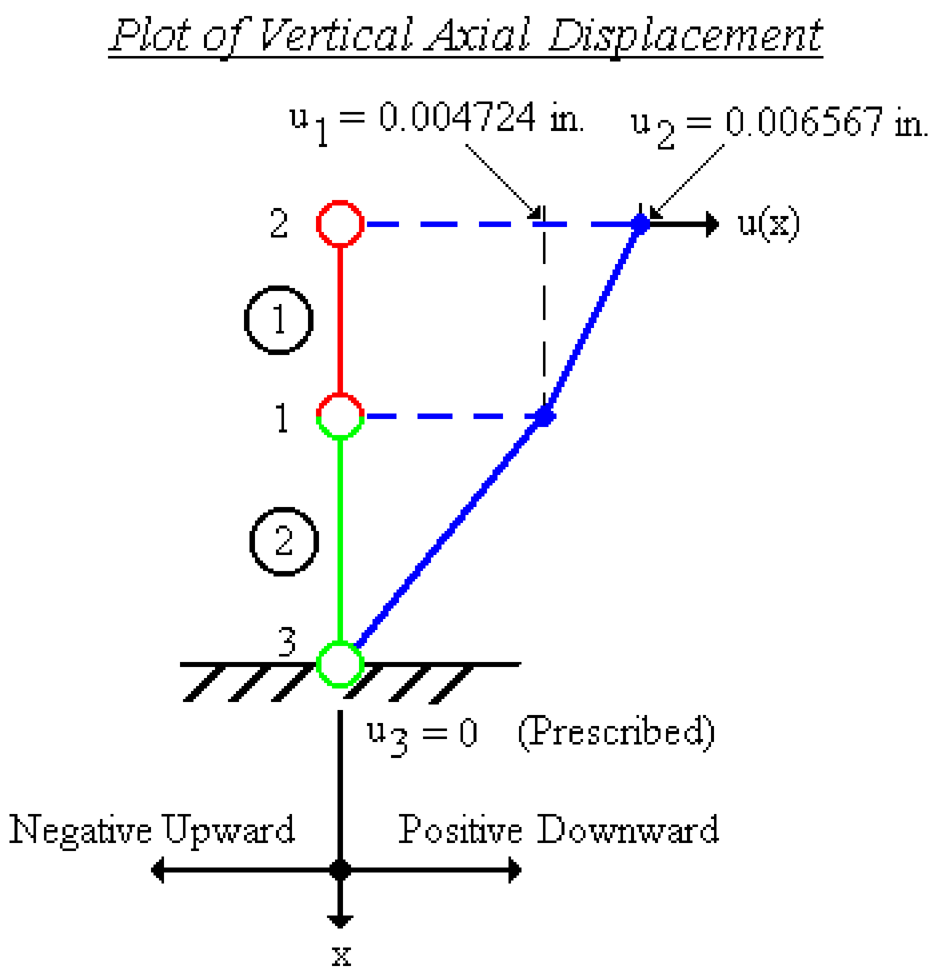
where

- b - known column vector of order n x 1.
- A - known square matrix of order n x n.
- x - unknown column vector of order n x 1.
- n - number of unknowns (number of nodes in this case).

$$\begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} 0.004724277 \text{ or } \mathbf{0.004724 \text{ in.}} \\ 0.006567081 \text{ or } \mathbf{0.006567 \text{ in.}} \end{Bmatrix}$$

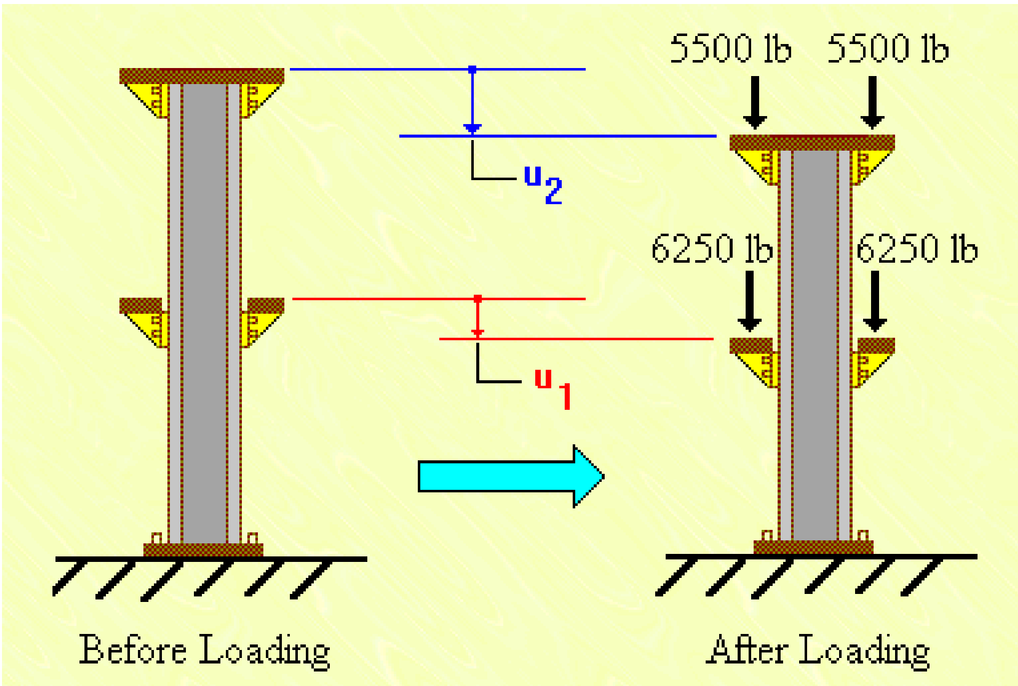
Nodal points 1 and 2 displace downward in accordance to positive element convention, as shown in the figure on the right-hand side. As a check of the nodal displacement u1 and u2, one can substitute them into the above equation and solve for F_1 and F_2.

3: Plot of axial displacement along the entire mesh.



The variation of the axial displacement along the structure can now be determined since all nodal displacements are known. The first step is to establish the coordinate system. In this case, since we want to present the vertical displacement along the entire structure, the coordinate system is constructed as shown on the left. Notice that the positive x-direction is downward, signifying the column length. Positive displacement which is plotted to the right signifies downward movement, and to the left denotes negative displacement which implies upward movement. The next step is to place the nodal displacements $u_1 = 0.004724$ in. and $u_2 = 0.006567$ in. at the nodes on the plot. In this case, both displacements are positive as they are both moving downward. According to the solution characteristic of the bar element the displacement variation is linear and continuous between the nodes. Therefore, we can simply connect them through a straight line.

Now the axial displacement of the structure can be seen clearly. Note that the slope of the two displacement curves are different, with the slope of element 2 being larger. This implies the change in length of element 2 is greater than that of element 1. However, as indicated from the plot, the maximum displacement occurs at node 2 which is part of element 1. The reason is because the displacement of element 1 is the sum of the deflection (deformation) of itself and also the rigid body motion from the deflection of element 2 at node 1. As a result, the total displacement (movement of each point of the element) would be greater than that of element 2. A physical illustration is shown below:



Mapping the original problem into the serial spring system at the beginning of this section yields the same results with:

$$k_1 = A\,E / L_2$$

$$k_2 = A\,E / L_1$$

$$P_1 = 1.25 * 10^4 \text{ lb}$$

$$P_2 = 1.1 * 10^4 \text{ lb}$$

$$u_1 = \delta_1 = (P_1 + P_2) / k_1$$

$$u_2 = \delta_2 = P_2 / k_2 + \delta_1$$

Importantly, note that the expressions for δ_1 and δ_2 were obtained previously by a work-energy minimization principle in addition to the force-based analysis.

ELEMENT INTERPOLATION FUNCTIONS and the INTEGRAL APPROACH IN FEM

Whereas the last FEM problem was clearly an exact solution within the assumptions of the model, the following 1-D heat conduction problem is solved only approximately. This is discussed in detail after a slightly more general treatment of element interpolation functions.

Extracted from **FEM NOTES** (1997)
by
Prof. Peter Hunter p.hunter@auckland.ac.nz
Andrew Pullan a.pullan@auckland.ac.nz

Contents