A First Course in Finite Elements

Introduction

The ***finite element method*** has become a powerful tool for the numerical solution of a wide range of engineering problems. Applications range from deformation and stress analysis of automotive, aircraft, building, and bridge structures to field analysis of heat flux, fluid flow, magnetic flux, seepage, and other flow problems.

With the advances in computer technology and CAD systems, complex problems can be modeled with relative ease. Several alternative configurations can be tried out on a computer before the first prototype is built. All of this suggests that we need to keep pace with these developments by understanding the basic theory, modeling techniques, and computational aspects of the finite element method.

In this method of analysis, a complex region defining a continuum is discretized into simple geometric shapes called ***finite elements***. The material properties and the governing relationships are considered over these elements and expressed in terms of unknown values at element corners. An assembly process, duly considering the loading and constraints, results in a set of equations. Solution of these equations gives us the approximate behavior of the continuum.

Historical Background

Basic ideas of the finite element method originated from advances in aircraft structural analysis. In 1941, Hrenikoff presented a solution of elasticity problems using the “frame work method.” Courant’s paper, which used piecewise polynomial interpolation over triangular subregions to model torsion problems, appeared in 1943. Turner et al. derived stiffness matrices for truss, beam, and other elements and presented their findings in 1956. The term finite element was first coined and used by Clough in 1960.

In the early 1960s, engineers used the method for approximate solution of problems in stress analysis, fluid flow, heat transfer, and other areas. A book by

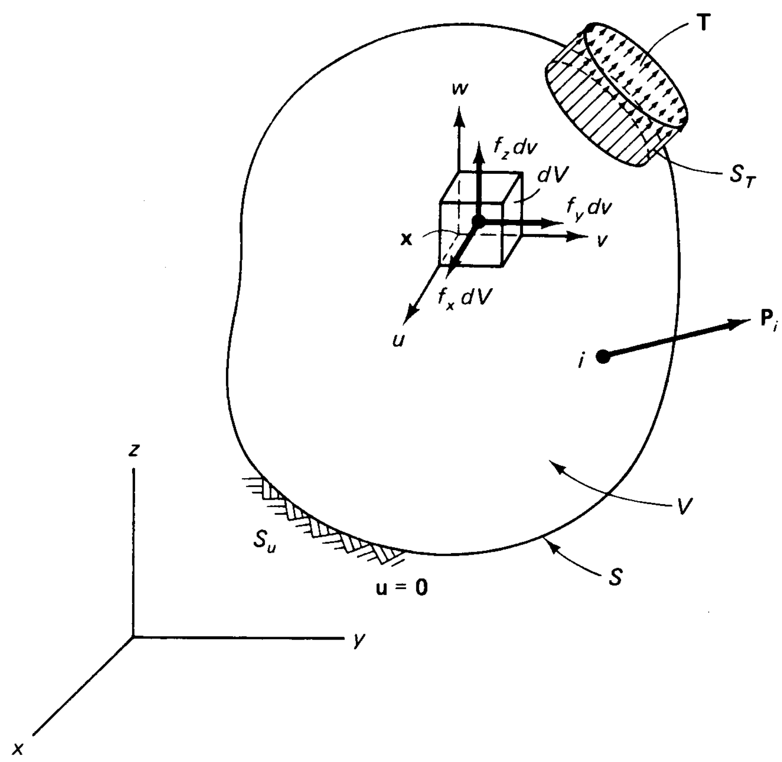
Argyris in 1955 on energy theorems and matrix methods laid a foundation for further developments in finite element studies. The first book on finite elements by Zienkiewicz and Chung was published in 1967. In the late 1960s and early 1970s, finite element analysis was applied to nonlinear problems and large deformations. Oden's book on nonlinear continua appeared in 1972.

Mathematical foundations were laid in the 1970s. New element development, convergence studies, and other related areas fall in this category.

Today, developments in mainframe computers and availability of powerful microcomputers have brought this method within reach of students and engineers working in small industries.

Stress and Equilibrium

Consider a three-dimensional body of volume ***V*** having a surface ***S***:



A point in the body is located by ***x, y***, and ***z*** coordinates. On part of the boundary, a distributed force per unit area ***T***, also called ***traction*** is applied. Under the force, the body deforms. The deformation of a point **x** (*x, y, z*) is given by the three components of its displacement vector:

$$\mathbf{u} = [u, v, w]^T$$

The distributed force per unit volume is given by force vector:

$$\mathbf{F} = [F_x, F_y, F_z]^T$$

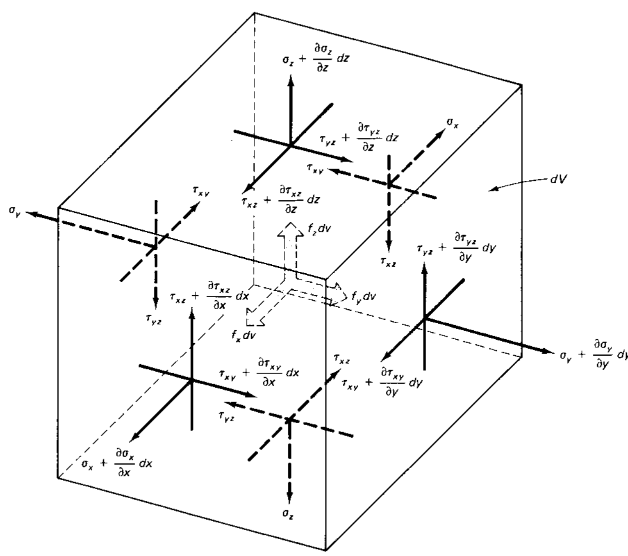
The traction ***T*** is given by its components at points along the surface:

$$\mathbf{T} = [T_x, T_y, T_z]^T$$

A load $\boldsymbol{P_i}$ acting at a point ***i*** is given by its three components:

$$\mathbf{P_i} = [P_x, P_y, P_z]_i^T$$

The stresses acting on the element volume ***dV*** are:



When the volume ***dV*** shrinks to a point, the stresses may be represented by placing its components in a (3 x 3) symmetric matrix. Stress can be represented by the six independent components:

$$\sigma = \left[\sigma_x \ \sigma_y \ \sigma_z \ \tau_{xy} \ \tau_{xz} \ \tau_{yz}\right]^T$$

where σ_x, σ_y, and σ_z are called ***normal stress*** and τ_{xy}, τ_{xz}, and τ_{yz} are called ***shear stress***. Consider the equilibrium of the element volume ***dV***. Forces are developed by multiplying the stresses by the corresponding areas. Writing the equations of equilibrium, recognizing the ***dV*** = *dx dy dz*:

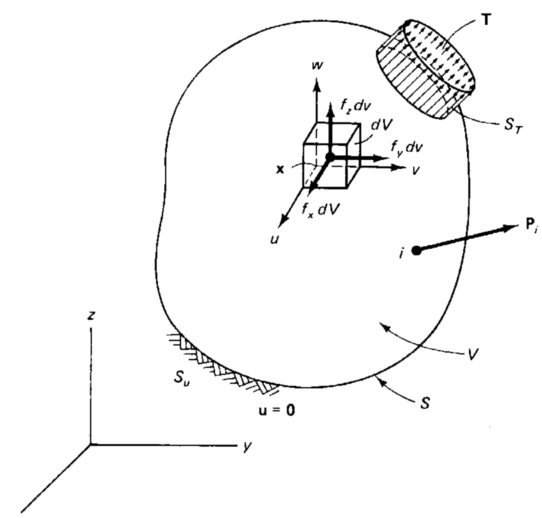
$$\sum F_x = \frac{\partial \sigma_x}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z} + f_x = 0$$

$$\sum F_y = \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \sigma_x}{\partial y} + \frac{\partial \tau_{yz}}{\partial z} + f_y = 0$$

$$\sum F_z = \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \sigma_z}{\partial z} + f_z = 0$$

Boundary Conditions

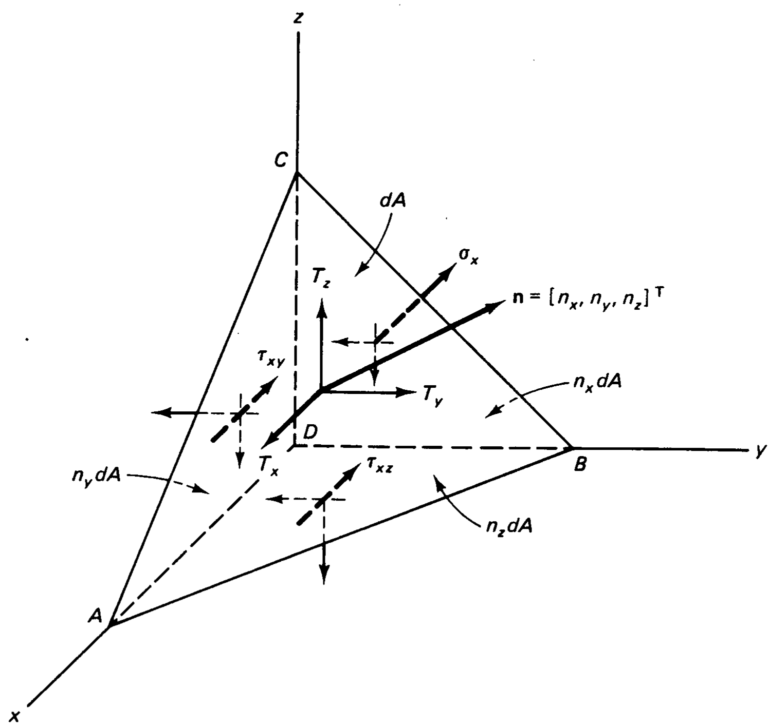
There are displacement boundary conditions and surface loading conditions.



u = c on Su

where c is a given displacement

Consider the equilibrium if an elemental tetrahedron ***ABCD***:



where ***DA***, ***DB***, and ***DC*** are parallel to the *x*, *y*, and *z* axes, respectively, and the area ***ABC***, denoted by ***dA***, lies on the surface. If the unit vector normal to the surface ***dA*** is given as:

$$\mathbf{n} = \left[n_x, n_y, n_z\right]^T$$

then the areas:

$$\mathbf{BDC} = n_x dA \qquad \mathbf{ADC} = n_y dA \qquad \mathbf{ADB} = n_z dA$$

Consider equilibrium in each direction:

$$\sigma_x n_x + \tau_{xy} n_y + \tau_{xz} n_z = T_x$$

$$\tau_{xy} n_x + \sigma_y n_y + \tau_{yz} n_z = T_y$$

$$\tau_{xz} n_x + \tau_{xy} n_y + \sigma_z n_z = T_y$$

These conditions must be satisfied on the boundary, $\boldsymbol{S_T}$, where the tractions are applied. Point loads must be treated as loads distributed over small but finite areas.

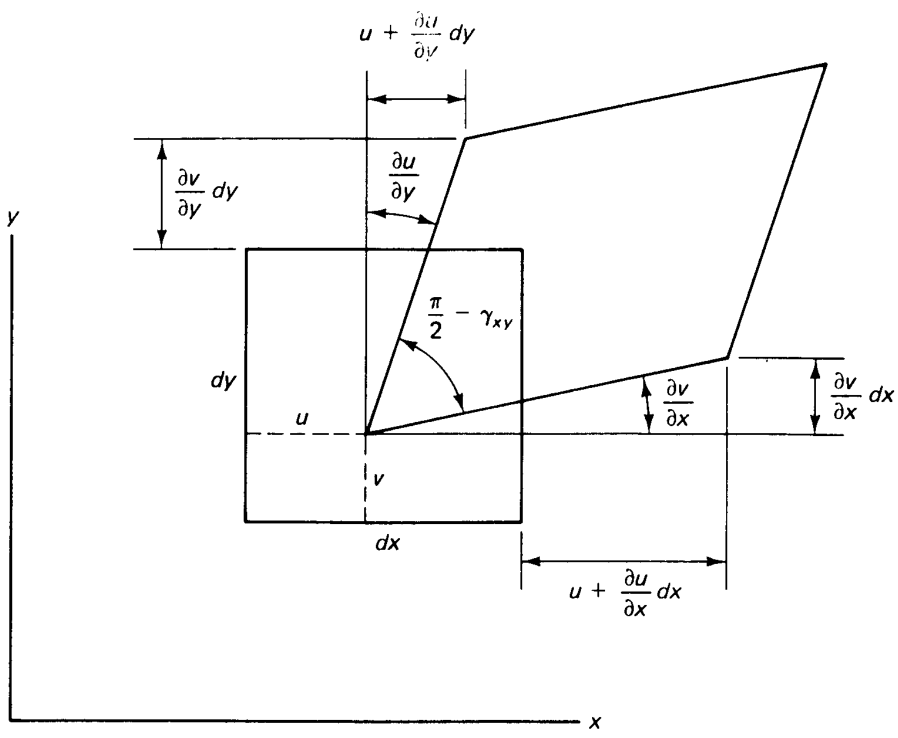
Strain-Displacement Relations

The strains in vector form that corresponds to the stress are:

$$\varepsilon = \left[\varepsilon_x, \varepsilon_y, \varepsilon_z, \gamma_{xy}, \gamma_{xz}, \gamma_{yz}\right]^T$$

where ε_x, ε_y, and ε_z are ***normal strains*** and γ_{xy}, γ_{xz}, and γ_{xz}, are the ***engineering shears strains***.

We can approximate the shear strains by considering a small deformation of the ***dx-dy*** face of the unit volume ***dV***:



$$\varepsilon = \left[\frac{\partial u}{\partial x}, \frac{\partial v}{\partial y}, \frac{\partial w}{\partial z}, \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}, \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}, \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}\right]^T$$

Stress-Strain Relations

For linear elastic materials, the stress-strain relations come from the generalized ***Hooke's Law***. For ***isotropic materials***, the two material properties are Young's modulus (or the modulus of elasticity) ***E*** and Poisson's ration ν. Considering an elemental volume inside the body, Hooke's Law gives:

$$\varepsilon_x = \frac{\sigma_x}{E} - \nu\frac{\sigma_y}{E} - \nu\frac{\sigma_z}{E}$$

$$\varepsilon_y = -\nu\frac{\sigma_x}{E} + \frac{\sigma_y}{E} - \nu\frac{\sigma_z}{E}$$

$$\varepsilon_z = -\nu\frac{\sigma_x}{E} - \nu\frac{\sigma_y}{E} + \frac{\sigma_z}{E}$$

$$\gamma_{yz} = \frac{\tau_{yz}}{G} \qquad \gamma_{xz} = \frac{\tau_{xz}}{G} \qquad \gamma_{xy} = \frac{\tau_{xy}}{G}$$

where the shear modulus (or modulus of rigidity), ***G***, is given by:

$$\mathbf{G} = \frac{E}{2(1+\nu)}$$

From Hooke's Law, strain and stress are related by:

$$\varepsilon_x + \varepsilon_y + \varepsilon_z = \frac{(1-2\nu)}{E}\left(\sigma_x + \sigma_y + \sigma_z\right)$$

By substituting the above relationships into Hooke's Law we get an inverse relationship:

$$\sigma = \mathbf{D}\varepsilon$$

where $\boldsymbol{D}$ is a symmetric (6 x 6) material matrix:

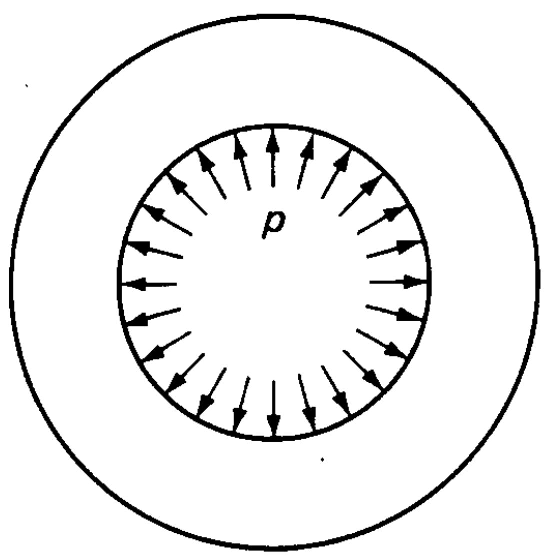
$$\mathbf{D} = \frac{E}{(1+\nu)(1-2\nu)}\begin{bmatrix} 1-\nu & \nu & \nu & 0 & 0 & 0 \\ \nu & 1-\nu & \nu & 0 & 0 & 0 \\ \nu & \nu & 1-\nu & 0 & 0 & 0 \\ 0 & 0 & 0 & 0.5-\nu & 0 & 0 \\ 0 & 0 & 0 & 0 & 0.5-\nu & 0 \\ 0 & 0 & 0 & 0 & 0 & 0.5-\nu \end{bmatrix}$$

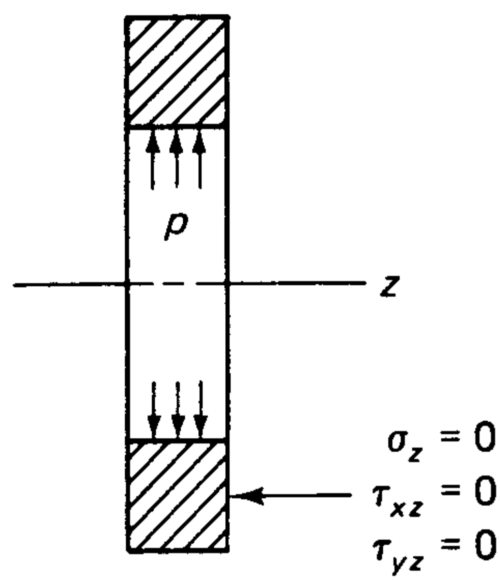
Special Cases

One-Dimension. In one-dimension, the normal stress s along ***x*** and the corresponding normal strain e. The stress-strain relationship is simply:

$$\sigma = \mathbf{E}\varepsilon$$

Two-Dimension. A thin planar body subjected to in-plane loading on its edge surface is said to be ***plane stress***. For example, consider a ring press fitted on a shaft:





Here the stresses σ_z, τ_{xz}, and τ_{yz} are assumed to be zero.

For plane stress, Hooke's Law reduces to:

$$\varepsilon_x = \frac{\sigma_x}{E} - \nu \frac{\sigma_y}{E}$$

$$\varepsilon_y = -\nu \frac{\sigma_x}{E} + \frac{\sigma_y}{E}$$

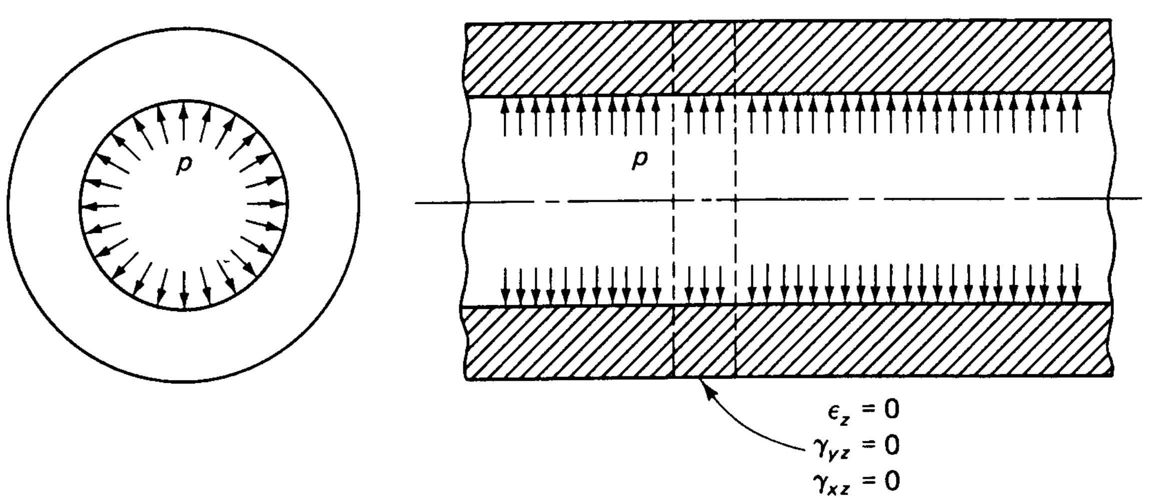
$$\varepsilon_z = -\frac{\nu}{E}\left(\sigma_x + \sigma_y\right)$$

$$\gamma_{xy} = \frac{2(1+\nu)}{E}\tau_{xy}$$

The inverse relationship σ **=** $\boldsymbol{D}\varepsilon$ reduces to:

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

If the body of uniform cross section is subjected to a transverse loading along its length, a small thickness in the loaded area can be approximated by ***plane strain***. For example:



The inverse relationship σ **=** ***D***ε reduces to:

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

Temperature Effects

If there is a temperature change Δ***T***(*x, y, z*) with respect to the original state, then an additional deformation can be estimated. For isotropic material, the temperature rise Δ***T*** results in a uniform strain; this depends on the coefficient of linear expansion of the material.

The temperature strain dose not cause any stresses when the body is free to deform. The temperature strain is represented as an initial strain:

$$\varepsilon_0 = [\alpha\Delta T, \alpha\Delta T, \alpha\Delta T, 0, 0, 0]^T$$

The stress-strain relationship becomes:

$$\sigma = D(\varepsilon - \varepsilon_{\mathbf{0}})$$

In ***plane stress***, we get:

$$\varepsilon_0 = [\alpha\Delta T, \alpha\Delta T, 0]^T$$

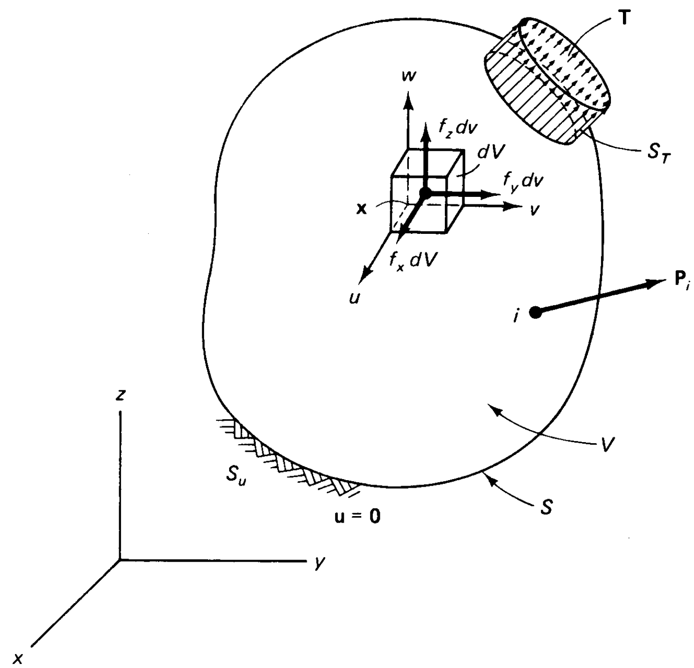
In ***plane strain***, we get:

$$\varepsilon_0 = (1+\nu)[\alpha\Delta T, \alpha\Delta T, 0]^T$$

For plane stress and plane strain σ, ε, and ***D*** are defined by the corresponding equations given above.

Potential Energy and Equilibrium; The Rayleigh- Ritz Method

In mechanics of solids, our problem is to determine the displacement ***u*** of the body, satisfying the equilibrium equations.



Note that stresses are related to strains, which, in turn, are related to displacements. This leads to requiring the solution of set of second-order partial differential equations. Solution of these equations is generally referred to as an ***exact solution***. Such exact solutions are available for simple geometries and loading conditions. For problems of complex geometries and general boundary and loading conditions, obtaining exact solutions is an almost impossible task. Approximate solution methods usually employ ***potential energy*** or ***variational methods***, which place less stringent conditions on the functions.

Potential Energy, Π

The total potential energy Π of an elastic body is defined as the sum of the total strain energy, ***U,*** and the work potential, ***WP***:

$$\Pi = \text{Strain Energy} + \text{Work Potential}$$
$$\qquad (U) \qquad\qquad\qquad (WP)$$

For linear elastic materials, the strain energy per unit volume is:

$$\frac{1}{2}\sigma^T\varepsilon$$

The total strain energy ***U*** is given as:

$$\mathbf{U} = \frac{1}{2}\int_V \sigma^T \varepsilon \, dV$$

The potential work ***WP*** is given as:

$$\mathbf{WP} = -\int_V \mathbf{u}^T \mathbf{f} \, dV - \int_S \mathbf{u}^T \mathbf{T} \, dS - \sum \mathbf{u}_i{}^T \mathbf{P}_i$$

The total potential energy for a general elastic body is:

$$\Pi = \frac{1}{2}\int_V \sigma^T \varepsilon \, dV - \int_V \mathbf{u}^T \mathbf{f} \, dV - \int_S \mathbf{u}^T \mathbf{T} \, dS - \sum \mathbf{u}_i{}^T \mathbf{P}_i$$

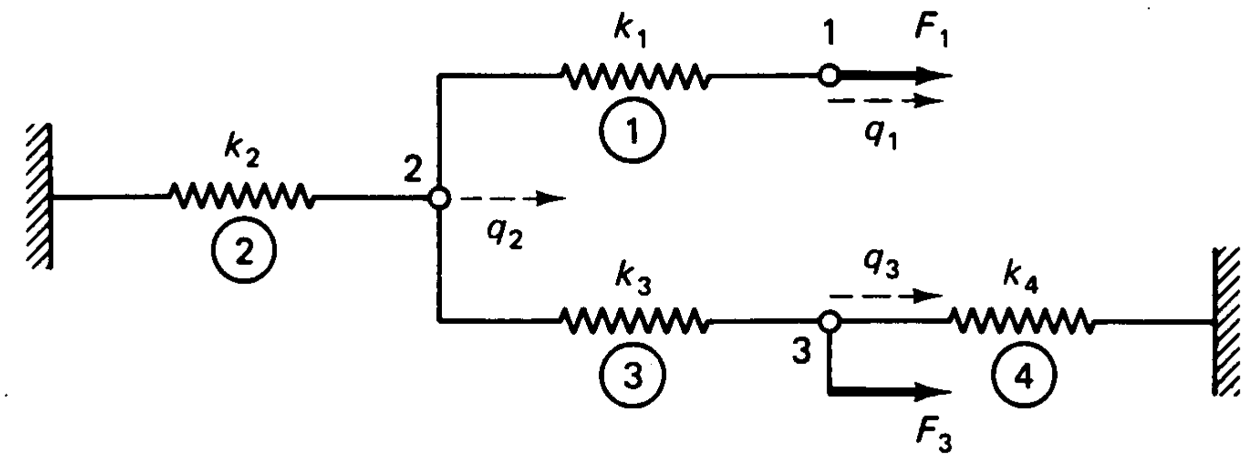
This is a conservative systems, where the work potential is independent of the path taken. In other words, if the system is displaced from a given configuration and brought back to its original state, the forces do zero work regardless of the path.

Principle of Minimum Potential Energy – For conservative systems, of all the kinematically admissible displacement fields, those corresponding to equilibrium extermize the total potential energy. If the extremum condition is a minimum, the equilibrium state is stable.

Kinematically admissible displacements are those that satisfy the single-valued nature of displacements (compatibility) and the boundary conditions.

Example

Consider a discrete connected system. The figure below shows a system of springs.



The total potential energy of the system is:

$$\Pi = \frac{1}{2}\left(k_1\delta_1^{\;2} + k_2\delta_2^{\;2} + k_3\delta_3^{\;2} + k_4\delta_4^{\;2}\right) - F_1q_1 - F_3q_3$$

where δ_1, δ_2, δ_3, and δ_4 are the extensions of the four springs.

$$\delta_1 = q_1 - q_2 \qquad \delta_2 = q_2$$

$$\delta_3 = q_3 - q_2 \qquad \delta_4 = -q_3$$

Therefore, total potential energy of the system is:

$$\Pi = \frac{1}{2}\left[k_1(q_1 - q_2)^2 + k_2q_2^{\;2} + k_3(q_3 - q_2)^2 + k_4q_3^{\;2}\right] - F_1q_1 - F_3q_3$$

where q_1, q_2, and q_3 are the displacements of nodes 1, 2, and 3, respectively.

For equilibrium of this three degree-of-freedom system, we need to minimize Π with respect to the displacements $\boldsymbol{q}_1$, $\boldsymbol{q}_2$, and $\boldsymbol{q}_3$.

$$\frac{\partial \Pi}{\partial q_1} = 0 \qquad i = 1, 2, 3$$

Therefore, the three equations are:

$$\frac{\partial \Pi}{\partial q_1} = k_1(q_1 - q_2) - F_1 = 0$$

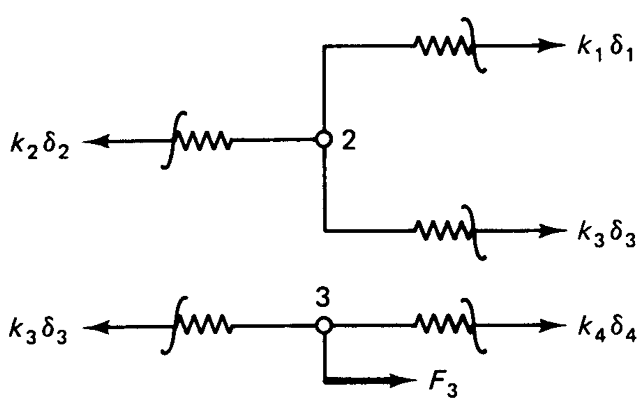
$$\frac{\partial \Pi}{\partial q_2} = -k_1(q_1 - q_2) + k_2 q_2 - k_3(q_3 - q_2) = 0$$

$$\frac{\partial \Pi}{\partial q_3} = k_3(q_3 - q_2) + k_4 q_3 - F_3 = 0$$

These equations can be written in matrix form as:

$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 + k_2 + k_3 & -k_3 \\ 0 & -k_3 & k_3 + k_4 \end{bmatrix} \begin{Bmatrix} q_1 \\ q_2 \\ q_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ 0 \\ F_3 \end{Bmatrix}$$

Alternately, we could write the equations of equilibrium for each node separately.



$$k_1\delta_1 = F_1$$

$$k_2\delta_2 - k_1\delta_1 - k_3\delta_3 = 0$$

$$k_3\delta_3 - k_4\delta_4 = F_3$$

Notice the equations for the displacements were obtained in a routine manner using the potential energy approach, without any reference to free body diagrams. This feature makes the potential energy approach attractive for large and complex problems.

Rayleigh-Ritz Method

For continua, the total potential energy, Π, can be used for finding an approximate solution. The Rayleigh-Ritz method involves the construction of an assumed displacement field [***u***, ***v***, ***w***]:

$$u = \sum a_i\phi_i(x, y, z) \quad i = 1 \; to \; l$$

$$v = \sum a_j\phi_j(x, y, z) \quad j = l + 1 \; to \; m$$

$$w = \sum a_k\phi_k(x, y, z) \quad k = m + 1 \; to \; n \qquad n > m > l$$

The functions ϕ_i are usually taken as polynomials. Displacements ***u***, ***v***, and ***w*** must be ***kinematically admissible*** (that is ***u***, ***v***, and ***w*** must satisfy boundary conditions). Introducing stress-strain and strain-displacement relationships gives:

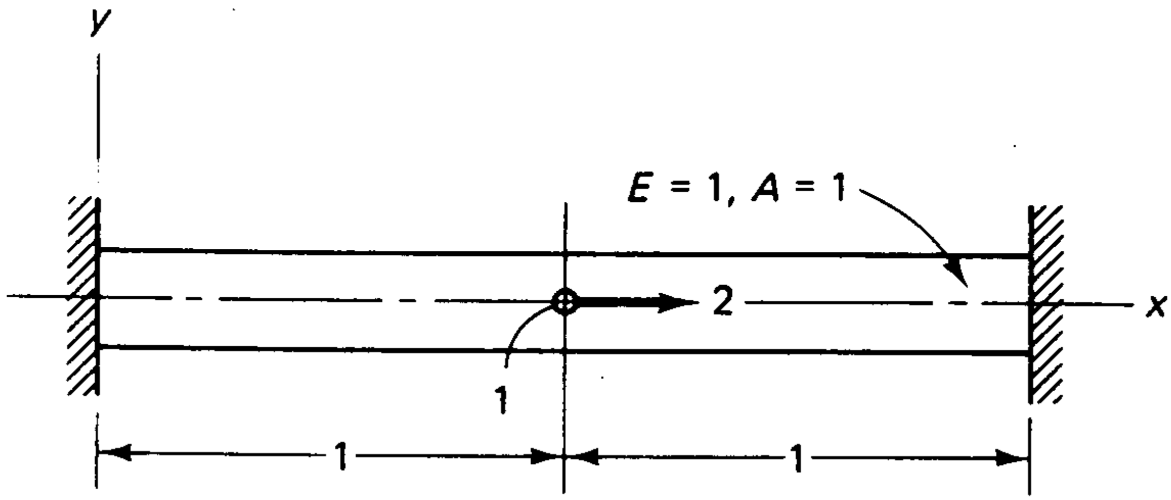
$$\Pi = \Pi(a_1, a_2, ..., a_n)$$

where ***n*** is the number of independent unknowns. The extremum with respect to $\boldsymbol{a_i}$, (i = 1 to n) gives a set of r equations:

$$\frac{\partial \Pi}{\partial a_i} = 0 \quad i = 1, 2, \cdots, n$$

Example

Consider the linear elastic one-dimensional rod with a body force shown below:



The potential energy of this system is:

$$\Pi = \frac{1}{2}\int_0^L EA\left(\frac{du}{dx}\right)^2 dx - 2u_1$$

where $\boldsymbol{u_1}$ = u(x=1). Consider the polynomial function:

$$u = a_1 + a_2 x + a_3 x^2$$

The kinematically admissible function $\boldsymbol{u}$ must satisfy the boundary conditions $\boldsymbol{u}$ = 0 at both (x = 0) and (x = 2). Therefore:

$$a_1 = 0 \qquad 2a_2 + 4a_3 = 0$$

Hence:

$$a_2 = -2a_3$$

$$u = a_3(-2x + x^2) \qquad u_1 = -a_3$$

$$\frac{du}{dx} = a_3 2(x-1)$$

The potential energy of this system using the function ***u*** is:

$$\Pi = \frac{1}{2}\int_0^2 EA4a_3^2(x-1)^2 dx + 2(a_3)$$

$$= 2EAa_3^2 \int_0^2 (1 - 2x + x^2)dx + 2a_3$$

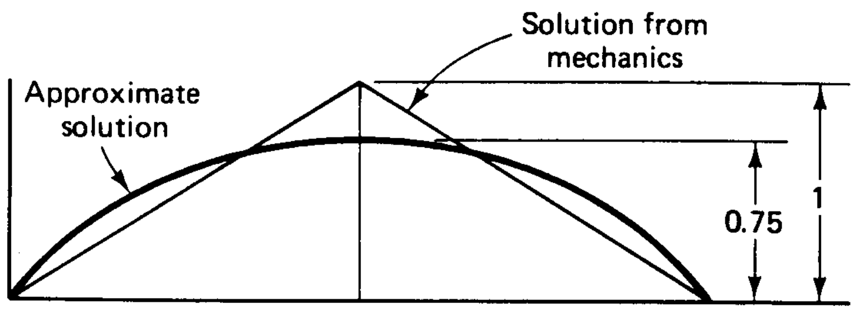
$$= \frac{4}{3}a_3^2 + 2a_3$$

Applying the Rayleigh-Ritz method gives:

$$\frac{\partial \Pi}{\partial a_3} = \frac{8}{3}a_3 + 2 = 0$$

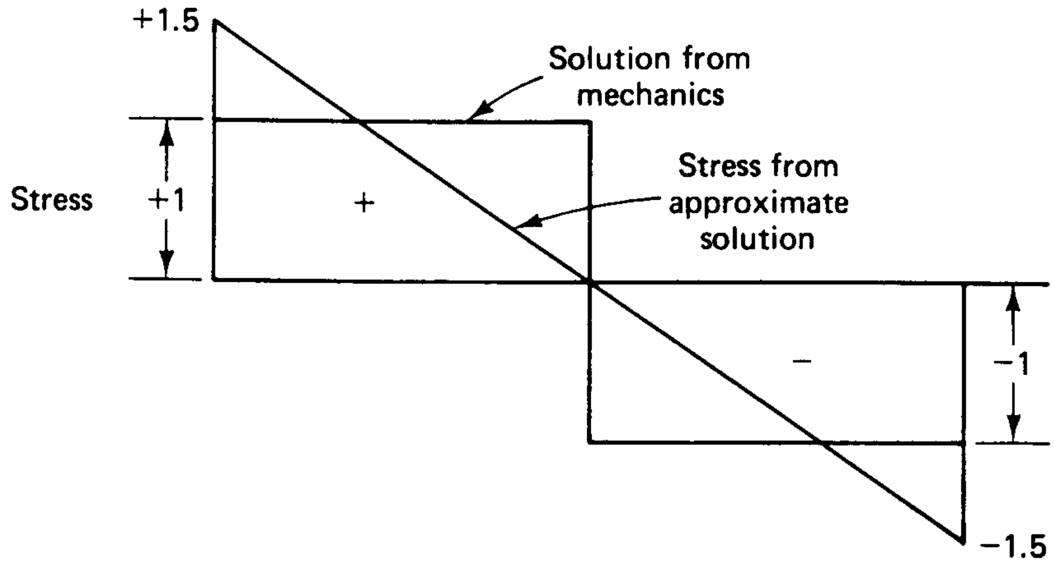
Solving for a_3 gives:

$$a_3 = -0.75 \qquad u_1 = -a_3 = 0.75$$



The stress in the bar is given by:

$$\sigma = E\frac{du}{dx} = 1.5(1-x)$$



Notice that an exact solution is obtained if a piecewise polynomial interpolation is used in the construction of ***u.***

Galerkin's Method

Galerkin's method uses the set of governing equations in the development of an integral form. It is usually presented as one of the weighted residual methods. For our discussion, let us consider a general representation of a governing equation on a region ***V***:

$$Lu = P$$

For the one-dimensional rod considered in the pervious example, the governing equation is:

$$\frac{d}{dx}\left(EA\frac{du}{dx}\right) = 0$$

If we consider ***L*** as the following operator:

$$\frac{d}{dx}EA\frac{d}{dx}()$$

operating on ***u***. The exact solution needs to satisfy ***L*** at every point ***x***. If we seek an approximate solution, $\hat{u}$, it introduces an error ***e(x)***, called the ***residual***:

$$e(x) = L\hat{u} - P$$

Approximate methods revolve around setting the residual relative to a weighting function $\boldsymbol{W_i}$, to zero.

$$\int_V W_i(Lu - P)dV = 0 \qquad i = 1 \; to \; n$$

The choice of the weighting function, $\boldsymbol{W_i}$, leads to various approximation methods. In the Galerkin methods, the weighting functions, $\boldsymbol{W_i}$, are chosen from the basis functions used for constructing $\hat{u}$. Let $\hat{u}$ be represented by:

$$\hat{u} = \sum_{i=1}^{n} Q_i G_i$$

where $\boldsymbol{G_i}$, i = 1 to n, are ***basis functions*** (usually polynomials of *x, y, z*). Here we choose the weighting function to a linear combination of the basis functions $\boldsymbol{G_i}$. Consider an arbitrary function f given by:

$$\phi = \sum_{i=1}^{n} \phi_i G_i$$

where the coefficients ϕ_i are arbitrary, except for requiring that ϕ satisfy homogeneous (zero) boundary conditions where $\hat{u}$ is prescribed.

Galerkin's Method – Chose basis functions G_i. Determine the coefficients Q_i such that

$$\int_V \phi(L\hat{u} - P)dV = 0 \quad \text{where } \phi = \sum_{i=1}^{n} \phi_i G_i$$

where ϕ_i are arbitrary except for requiring that ϕ satisfy homogeneous boundary conditions. The solution of the resulting equations for Q_i then yields the approximate solution $\hat{u}$.

Galerkin's Method in Elasticity

Consider the equations of equilibrium we developed earlier. Galerkin's method requires:

$$\int_V \left[\left(\frac{\partial \sigma_x}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z} + f_x \right)\phi_x + \left(\frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \sigma_y}{\partial y} + \frac{\partial \tau_{yz}}{\partial z} + f_y \right)\phi_y \right.$$

$$\left. + \left(\frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \sigma_z}{\partial z} + f_z \right)\phi_z \right] dV = 0$$

where

$$\phi = \left[\phi_x, \phi_y, \cdots \phi_z \right]^T$$

is an arbitrary displacement consistent with the displacements, ***u***. Consider integration by parts using the following formula:

$$\int_V \frac{\partial \alpha}{\partial x} \theta \, dV = -\int_{\partial_V} \alpha \frac{\partial \theta}{\partial x} dV + n_x \alpha \theta \, dS$$

where α and θ are functions of (x, y, z). For multi-dimensional problems the above equation is referred to as Green-Gauss theorem or the divergence theorem.

Using the Green-Gauss theorem on the equations of equilibrium yields:

$$-\int_V \sigma^T \varepsilon(\phi)\, dV + \int_V \phi^T f\, dV$$

$$+\int_S (n_x\sigma_x + n_y\tau_{xy} + n_z\tau_{xz}) f_x\, dS$$

$$+\int_S (n_x\tau_{xy} + n_y\sigma_y + n_z\tau_{yz}) f_y\, dS$$

$$+\int_S (n_x\tau_{xz} + n_y\tau_{yz} + n_z\sigma_z) f_z\, dS$$

where $\varepsilon(\phi)$ is the strain field corresponding to the arbitrary displacement field ϕ.

$$\varepsilon(\phi) = \left[\frac{\partial\phi_x}{\partial x}, \frac{\partial\phi_y}{\partial y}, \frac{\partial\phi_z}{\partial z}, \frac{\partial\phi_y}{\partial z} + \frac{\partial\phi_z}{\partial y}, \frac{\partial\phi_x}{\partial z} + \frac{\partial\phi_z}{\partial x}, \frac{\partial\phi_x}{\partial y} + \frac{\partial\phi_y}{\partial x}\right]^T$$

On the boundary we have:

$$\sigma_x n_x + \tau_{xy} n_y + \tau_{xz} n_z = T_x$$

$$\tau_{xy} n_x + \sigma_y n_y + \tau_{yz} n_z = T_y$$

$$\tau_{xz} n_x + \tau_{xy} n_y + \sigma_z n_z = T_y$$

At a point loads:

$$\left(\sigma_x n_x + \tau_{xy} n_y + \tau_{xz} n_z\right) dS = P_x$$

$$\left(\tau_{xy} n_x + \sigma_y n_y + \tau_{yz} n_z\right) dS = P_y$$

$$\left(\tau_{xz} n_x + \tau_{xy} n_y + \sigma_z n_z\right) dS = P_y$$

These are the natural boundary conditions in the problem. Therefore the Galerkin "weak form" or "variational form" for three-dimensional stress analysis is:

$$\int_V \sigma^T \varepsilon(\phi)\, dV - \int_V \phi^T \mathbf{f}\, dV - \int_S \phi^T \mathbf{T}\, dS - \sum \phi^T \mathbf{P} = 0$$

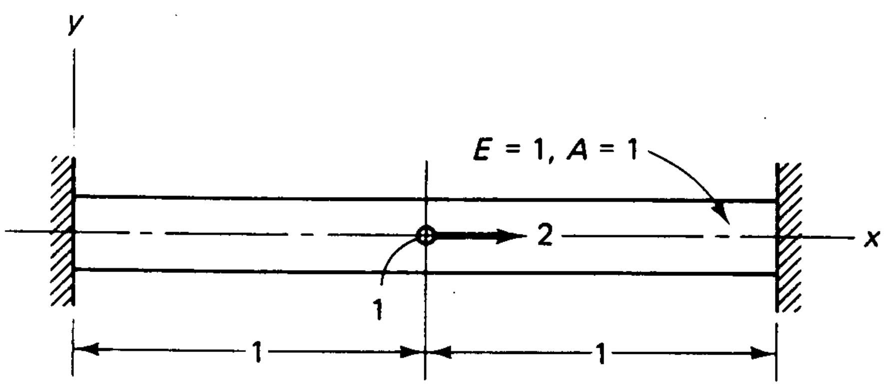
where ϕ is an arbitrary displacement consistent with the boundary conditions. For problems of linear elasticity, the above equation is precisely the ***principle of virtual work***. The function ϕ is the admissible virtual displacement. The principle of virtual work may be stated as follows:

> **Principle of Virtual Work** – A body is in equilibrium if the internal work equals the external virtual work for every kinematically admissible displacement field.

Example

Let consider the pervious problem and solve it by Galerkin's approach. The equilibrium equation is:

$$\frac{d}{dx} EA \frac{du}{dx} = 0 \qquad u = 0 \text{ at } x = 0 \qquad u = 0 \text{ at } x = 2$$



Multiplying the differential equation above by ϕ and integrating by parts gives:

$$-\int_1^2 EA \frac{du}{dx}\frac{d\phi}{dx} dx + \left(\phi EA \frac{du}{dx}\right)_0^1 + \left(\phi EA \frac{du}{dx}\right)_1^2 = 0$$

where ϕ is zero at (***x*** = 0) and (*x* = 2) and *EA(du/dx)* is the tension in the rod, which make a jump of magnitude of 2 at (*x* = 1). Therefore:

$$-\int_0^2 EA\frac{du}{dx}\frac{d\phi}{dx}dx + 2\phi_1 = 0$$

If we use the same polynomial function for ***u*** and ϕ and If $\boldsymbol{u}_1$ and ϕ_1 are the values at (x = 1), we get:

$$u = (2x - x^2)u_1 \qquad \phi = (2x - x^2)\phi_1$$

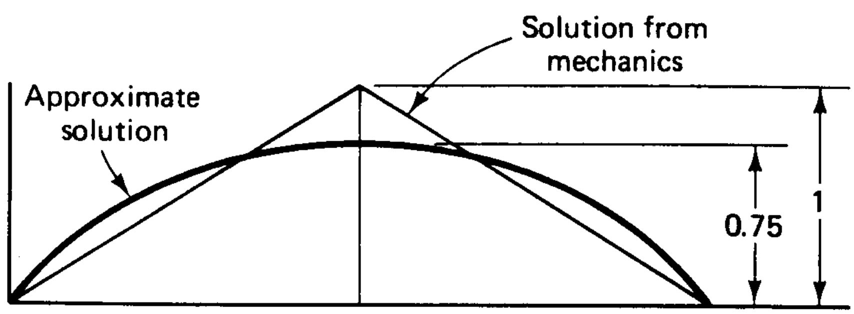
Substituting these and E = A = 1 in the above integral:

$$\phi_1\left[-u_1\int_0^2 (2-2x)^2 dx + 2\right] = 0$$

$$\phi_1\left(-\frac{8}{3}u_1 + 2\right) = 0$$

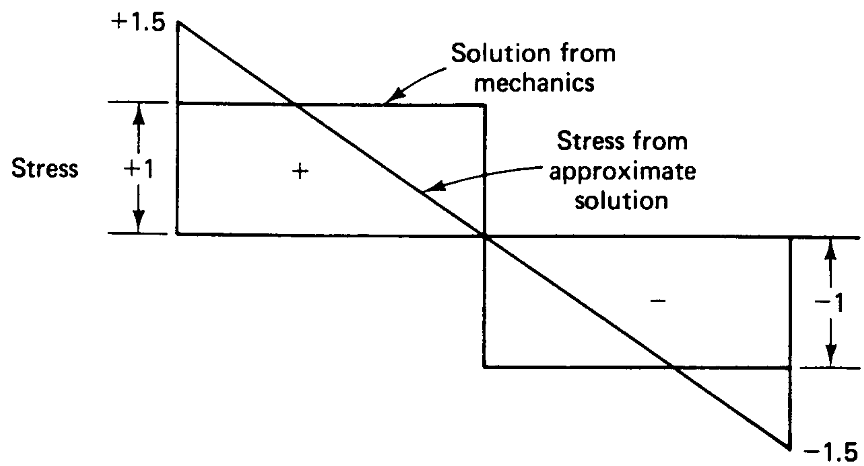
This is to be satisfied for every ϕ_1. We get:

$$u_1 = 0.75$$



The stress in the bar is given by:

$$\sigma = E\frac{du}{dx} = 1.5(1-x)$$

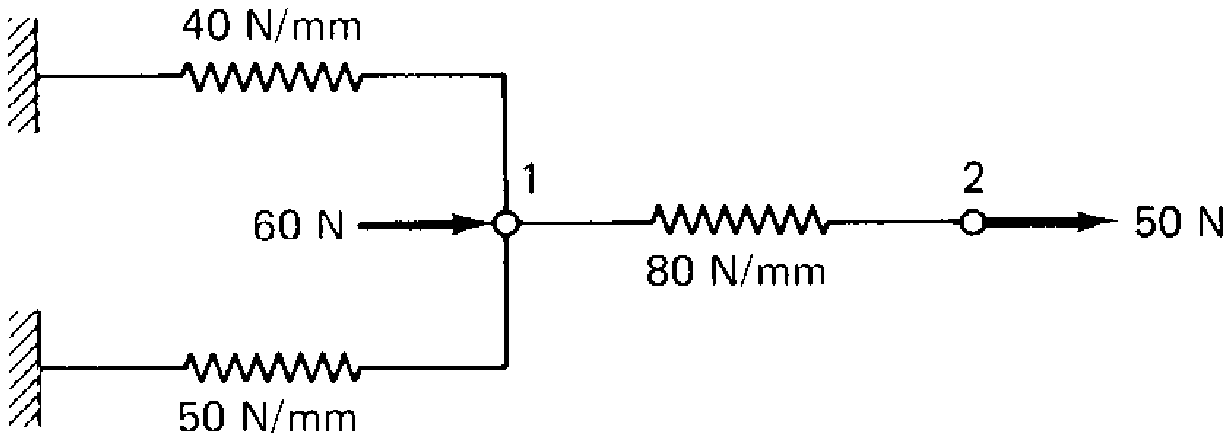


Problems:

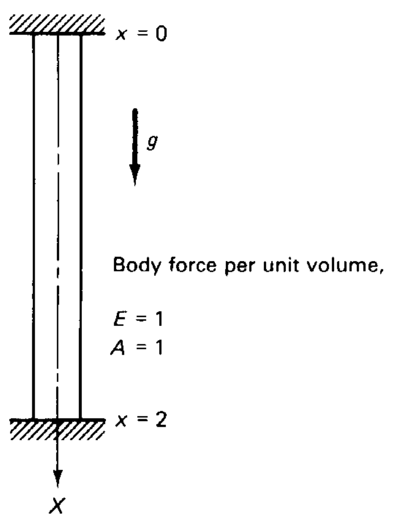
1. Obtain the **D** matrix given below using the generalized Hook's law relationships.

$$\mathbf{D} = \frac{E}{(1+\nu)(1-2\nu)}\begin{bmatrix} 1-\nu & \nu & \nu & 0 & 0 & 0 \\ \nu & 1-\nu & \nu & 0 & 0 & 0 \\ \nu & \nu & 1-\nu & 0 & 0 & 0 \\ 0 & 0 & 0 & 0.5-\nu & 0 & 0 \\ 0 & 0 & 0 & 0 & 0.5-\nu & 0 \\ 0 & 0 & 0 & 0 & 0 & 0.5-\nu \end{bmatrix}$$

2. Determine the displacement of nodes of the spring system shown below:



3. Use the Rayleigh-Ritz method to find the displacement of the midpoint of the rod shown below (assume ρg = 1):



4. Use Galerkin's method to find the displacement of the midpoint of the rod in Problem 3.

Role of Computers in Finite Element Methods

Until the early 1950s, matrix methods and the associated finite element method were not readily adaptable for solving complicated problems because of the large number of algebraic equations that resulted. Hence, even though the finite element method was being used to describe complicated structures, the resulting large number of equations associated with the finite element method of structural analysis made the method extremely difficult and impractical to use.

With the advent of the computer, the solution of thousands of equations in a matter of minutes became possible. The development of the computer resulted in computational program development. Numerous special-purpose and general-purpose programs have been written to handle various complicated structural (and non-structural) problems. To use the computer, the analyst, having defined the finite element model, inputs the information into the computer. This formation may include the position of the element nodal coordinates, the manner in which elements are connected together, the material properties of the elements, the applied loads, boundary conditions, or constraints, and the kind of analysis to be performed. The computer then uses this information to generate and solve the equations necessary to carry out the analysis.

General Steps of the Finite Element Method

The following section presents the general steps for applying the finite element method to obtain solutions of structural engineering problem. Typically, for the structural stress-analysis problem, the engineer seeks to determine ***displacements*** and ***stresses*** throughout the structure, which is in equilibrium and is subjected to applied loads. For many structures, it is difficult to determine the distribution of deformation using conventional methods, and thus the finite element method is necessarily used.

There are two general approaches associated with the finite element method. One approach, called the ***force***, or ***flexibility method***, uses internal forces as the unknowns of the problem. To obtain the governing equations, first the equilibrium equations are used. Then necessary additional equations are found by introducing compatibility equations. The result is a set of algebraic equations for determining the redundant or unknown forces.

The second approach, called the ***displacement***, or ***stiffness method***, assumes the displacements of the nodes as the unknowns of the problem. The governing equations are expressed in terms of nodal displacements using the equations of equilibrium and an applicable law relating forces to displacements.

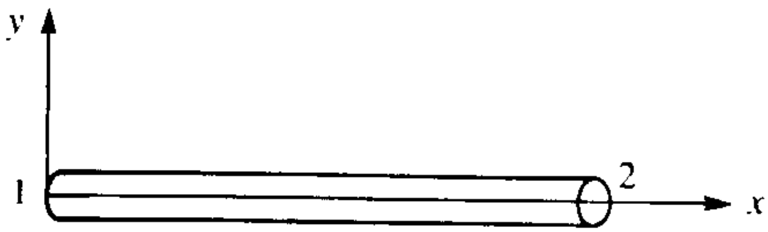
These two approaches result in different unknowns (forces or displacements) in the analysis and different matrices associated with their formulations (flexibilities or stiffnesses). It has been shown that, for computational purposes, the displacement (or stiffness) method is more desirable because its formulation is simpler for most structural analysis problems. Consequently, only the displacement method will be used throughout this text.

The ***finite element method*** involves modeling the structure using small interconnected elements called ***finite elements***. A displacement function is associated with each finite element. Every interconnected element is linked, directly or indirectly, to every other element through common (or shared) interfaces, including nodes and/or boundary lines and/or surfaces. The total set of equations describing the behavior of each node results in a series of algebraic equations best expressed in matrix notation.

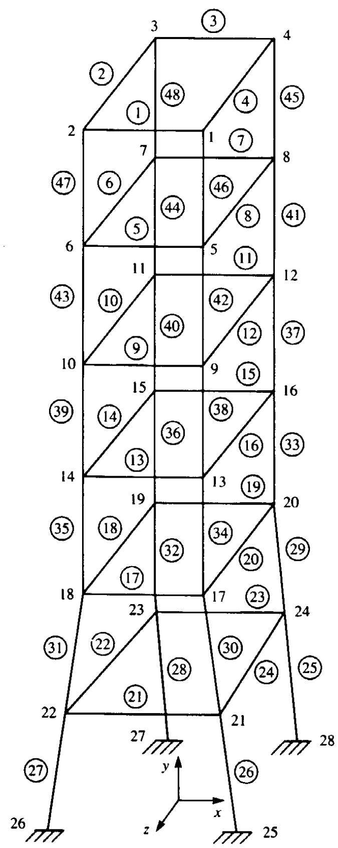
Step 1 - Discretize and Select Element Types

Step 1 involves dividing the body into an equivalent system of finite elements with associated nodes and choosing the most appropriate element type. The total number of elements used and their variation in size and type within a given body are primarily matters of engineering judgment. The elements must be made small enough to give usable results and yet large enough to reduce computational effort. Small elements (and possibly higher-order elements) are generally desirable where the results are changing rapidly, such as where changes in geometry occur, whereas large elements can be used where results are relatively constant.

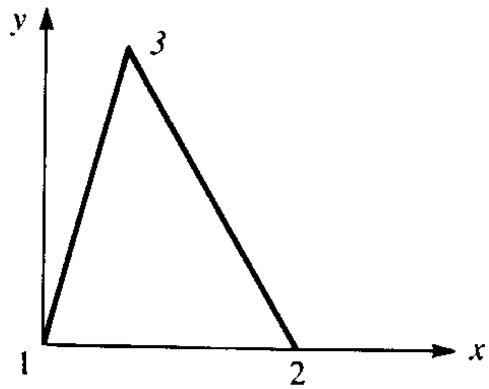
The primary ***line elements***, consist of bar (or truss) and ***beam elements***. They have a cross-sectional area but are usually represented by line segments. In general, the cross-sectional area within the element can vary, but it will be considered to be constant throughout this text.

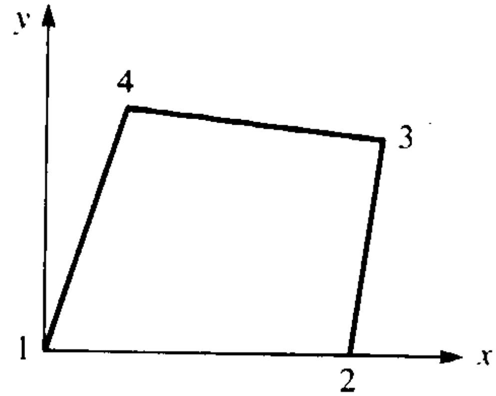


These elements are often used to model trusses and frame structures. The simplest line element (called a ***linear element***) has two nodes, one at each end, although higher-order elements having three nodes or more (called ***quadratic***, ***cubic***, etc. ***elements***) also exist. The line elements are the simplest of elements to consider and will be used to illustrate many of the basic concepts of the finite element method.



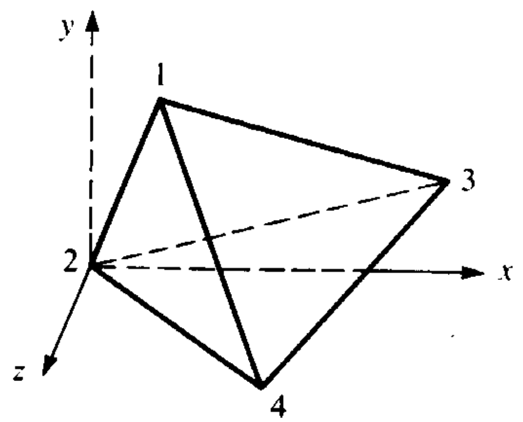
The basic two-dimensional (or plane) elements are loaded by forces in their own plane (plane stress or plane strain conditions). They are triangular or quadrilateral elements.

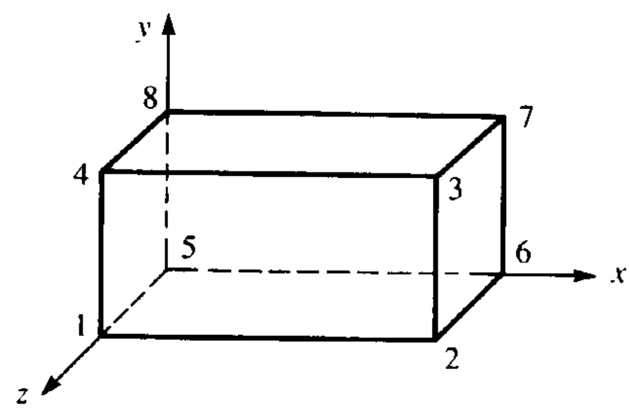


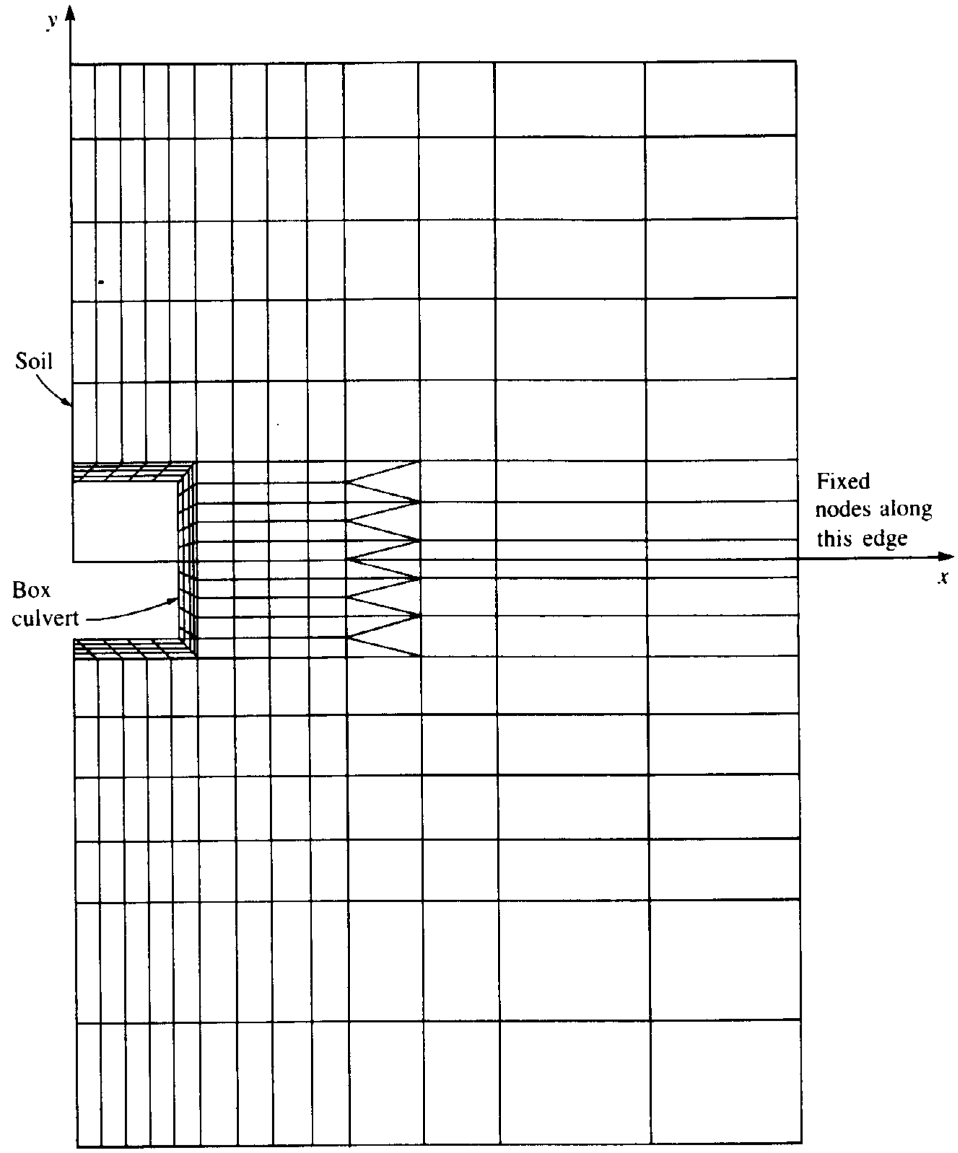


The simplest two-dimensional elements have corner nodes only (linear elements) with straight sides or boundaries although there are also higher-order elements, typically with mid-side nodes (called ***quadratic elements***) and curved sides. The elements can have variable thicknesses throughout or be constant. They are often used to model a wide range of engineering problems.

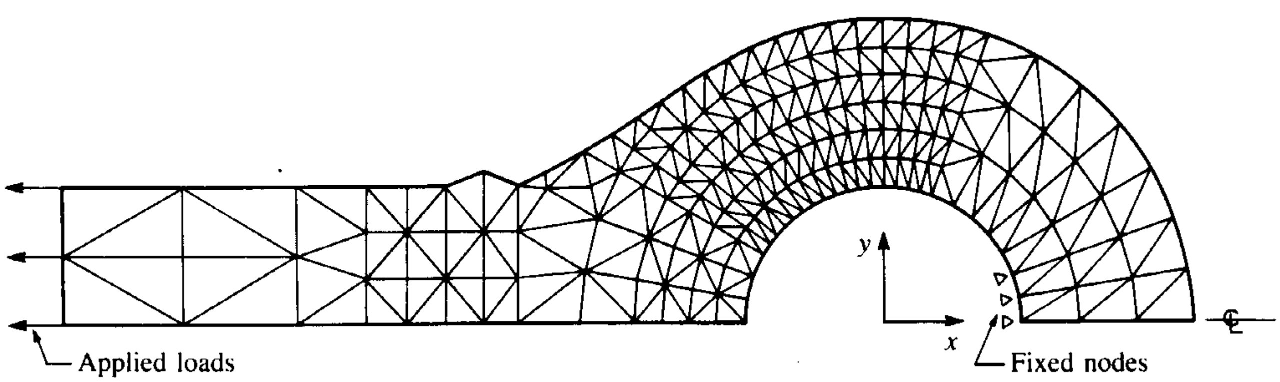
The most common three-dimensional elements are tetrahedral and hexahedral (or ***brick***) ***elements***; they are used when it becomes necessary to perform a three-dimensional stress analysis. The basic three dimensional elements have corner nodes only and straight sides, whereas higher-order elements with mid-edge nodes (and possible mid-face nodes) have curved surfaces for their sides.



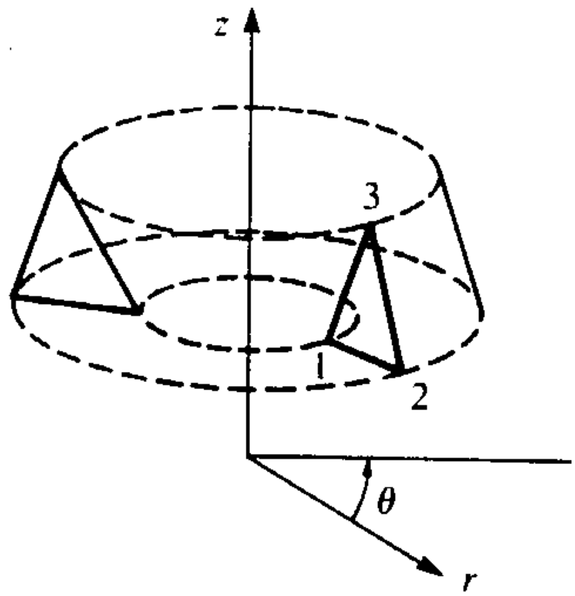




Fixed nodes along bottom



The ***axisymmetric element*** is developed by rotating a triangle or quadrilateral about a fixed axis located in the plane of the element through 360°. This element can be used when the geometry and loading of the problem are axisymmetric



Step 2 - Select a Displacement Function

Step 2 involves choosing a displacement function within each element. The function is defined within the element using the nodal values of the element. Linear, quadratic, and cubic polynomials are frequently used functions because they are simple to work with in finite element formulation. The functions are expressed in terms of the nodal unknowns (in the two-dimensional problem, in terms of an *x* and a *y* component). Hence, the finite element method is one in which a continuous quantity, such as the displacement throughout the body, is approximated by a discrete model composed of a set of piecewise-continuous functions defined within each finite domain or finite element.

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

Strain/displacement and stress-strain relationships are necessary for deriving the equations for each finite element. For one-dimensional small strain deformation, say, in the *x* direction, we have strain ε_x, related to displacement ***u*** by:

$$\varepsilon_x = \frac{du}{dx}$$

In addition, the stresses must be related to the strains through the stress-strain law (generally called the ***constitutive law***). The ability to define the material behavior accurately is most important in obtaining acceptable results. The simplest of stress-strain laws, Hooke's law, often used in stress analysis, is given by:

$$\sigma_x = E\varepsilon_x$$

Step 4 - Derive the Element Stiffness Matrix and Equations

Initially, the development of element stiffness matrices and element equations was based on the concept of stiffness influence coefficients, which presupposes a background in structural analysis. We now present alternative methods used in this text that do not require this special background.

Direct Equilibrium Method - According to this method, the stiffness matrix and element equations relating nodal forces to nodal displacements are obtained using force equilibrium conditions for a basic element, along with force-deformation relationships. This method is most easily adaptable to line or one-dimensional elements (spring, bar, and beam elements).

Work or Energy Methods - To develop the stiffness matrix and equations for two- and three-dimensional elements, it is much easier to apply a work or energy method. The ***principle of virtual work*** (using virtual displacements), the principle of minimum potential energy, and Castigliano's theorem are methods frequently used for the purpose of derivation of element equations. We will present the principle of minimum potential energy (probably the most well known of the three energy methods mentioned here).

Methods of Weighted Residuals - The methods of weighted residuals are useful for developing the element equations (particularly popular is Galerkin's method). These methods yield the same results as the energy methods, wherever the energy methods are applicable. They are particularly useful when a ***functional*** such as potential energy is not readily available. The weighted residual methods allow the finite element method to be applied directly to any differential equation.

Step 5 - Assemble the Element Equations and Introduce Boundary Conditions

The individual element equations generated in Step 4 can now be added together using a method of superposition (called the ***direct stiffness method***) whose basis is nodal force equilibrium (to obtain the global equations for the whole structure). Implicit in the direct stiffness method is the concept of continuity, or compatibility, which requires that the structure remain together and that no tears occur anywhere in the structure. The final assembled or global equation written in matrix form is:

$$\{F\} = [K]\{d\}$$

where **{*F*}** is the vector of global nodal forces, **[*K*]** is the structure global or total stiffness matrix, and **{*d*}** is now the vector of known and unknown structure nodal degrees of freedom or generalized displacements.

Step 6 - Solve for the Unknown Degrees of Freedom (or Generalized Displacements)

Once the element equations are assembled and modified to account for the boundary conditions, a set of simultaneous algebraic equations that can be written in expanded matrix form as:

$$\begin{Bmatrix} F_1 \\ F_2 \\ \cdot \\ F_n \end{Bmatrix} = \begin{bmatrix} K_{11} & K_{12} & \cdot & K_{1n} \\ K_{21} & K_{22} & \cdot & K_{2n} \\ \cdot & \cdot & \cdot & \cdot \\ K_{n1} & K_{n2} & \cdot & K_{nn} \end{bmatrix} \begin{Bmatrix} d_1 \\ d_2 \\ \cdot \\ d_n \end{Bmatrix}$$

where ***n*** is the structure total number of unknown nodal degrees of freedom. These equations can be solved for the ***d***'s by using an elimination method (such as Gauss's method) or an iterative method (such as Gauss Seidel's method).

Step 7 - Solve for the Element Strains and Stresses

For the structural stress-analysis problem, important secondary quantities of strain and stress (or moment and shear force) can be obtained in terms of the displacements determined in Step 6.

Step 8 - Interpret the Results

The final goal is to interpret and analyze the results for use in the design/analysis process. Determination of locations in the structure where large deformations and large stresses occur is generally important in making design/analysis decisions. Post-processor computer programs help the user to interpret the results by displaying them in graphical.

Applications of the Finite Element Method

The following applications will illustrate the variety, size, and complexity of problems that can be solved using the finite element method and the typical discretization process and kinds of elements used.

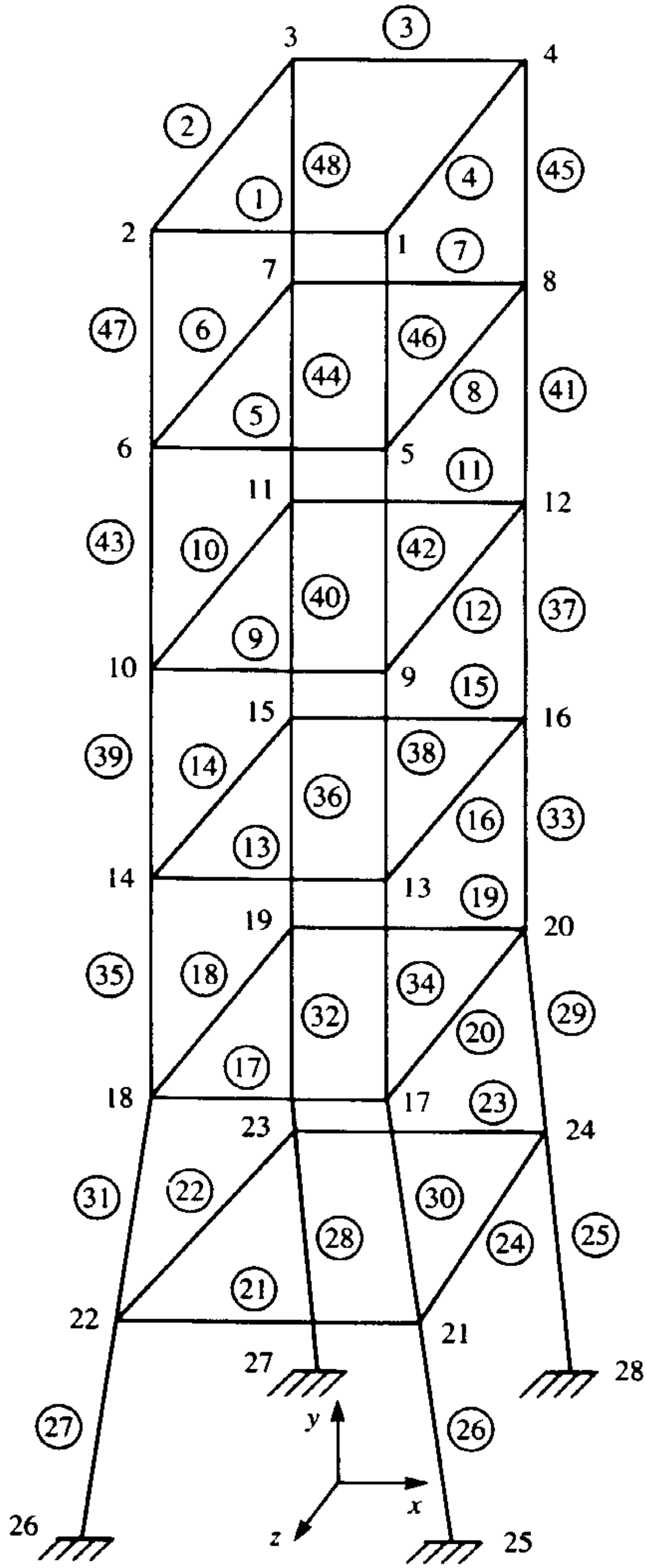
The first example is of a control tower for a railroad. The tower is a three-dimensional frame comprising a series of beam-type elements. The 48 elements are labeled by the circled numbers, whereas the 28 nodes are indicated by the encircled numbers. Each node has three rotation and three displacement components associated with it. The rotations and displacements are called the ***degrees of freedom***.

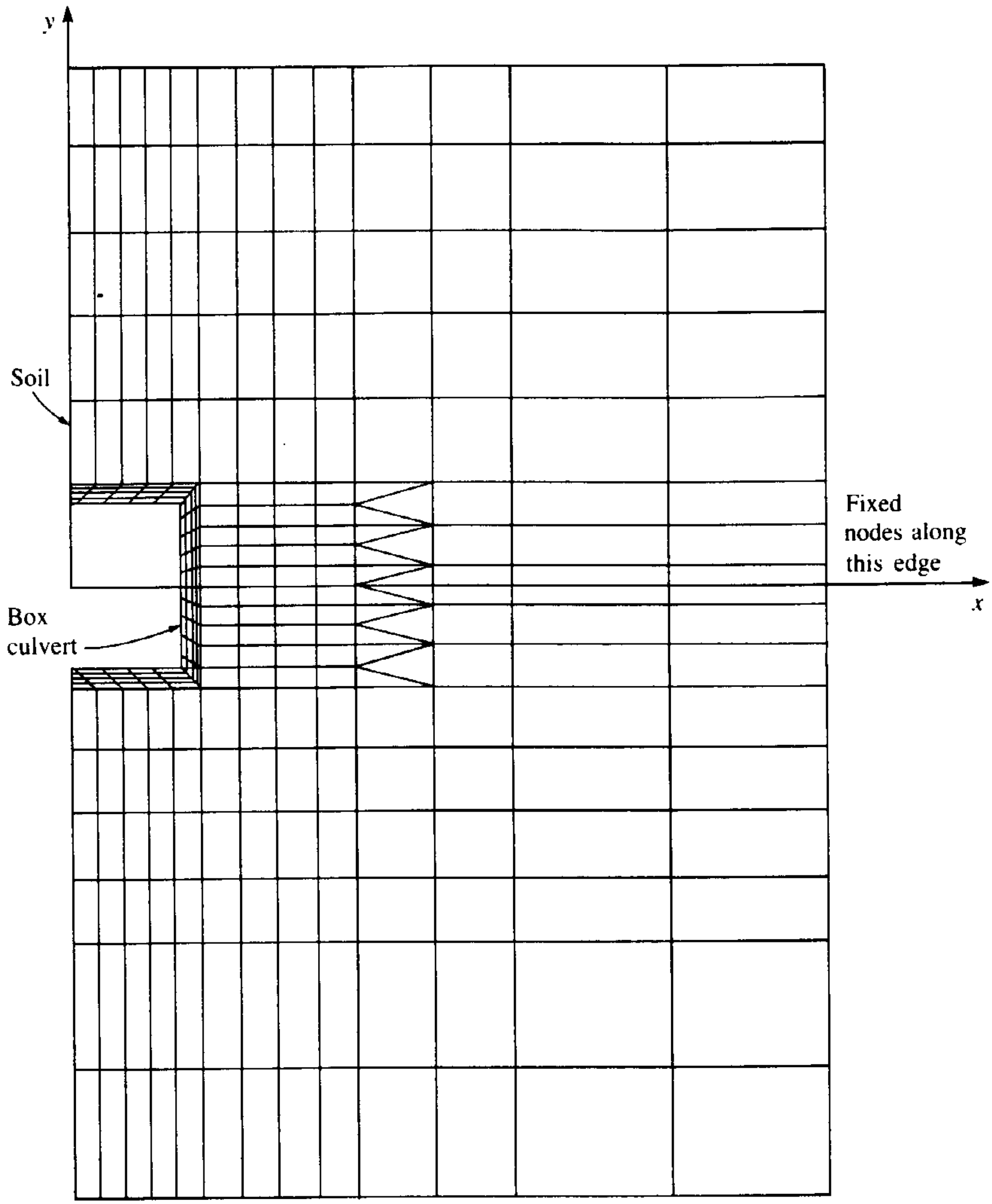
The next example is the determination of displacements and stresses in an underground box culvert subjected to ground shock loading from a bomb explosion. The discretized model that included a total of 369 nodes, 40 one-dimensional bar or truss elements used to model the steel reinforcement in the box culvert, and 333 plane strain two-dimensional triangular and rectangular elements used to model the surrounding soil and concrete box culvert. With an assumption of symmetry, only half of the box culvert must be analyzed. This problem requires the solution of nearly 700 unknown nodal displacements.

Another two-dimensional problem is that of a hydraulic cylinder rod end. It was modeled by 120 nodes and 297 plane strain triangular elements. Symmetry was also applied to the whole rod end so that only half of the rod end had to be analyzed, as shown. The purpose of this analysis was to locate areas of high stress concentration in the rod end.

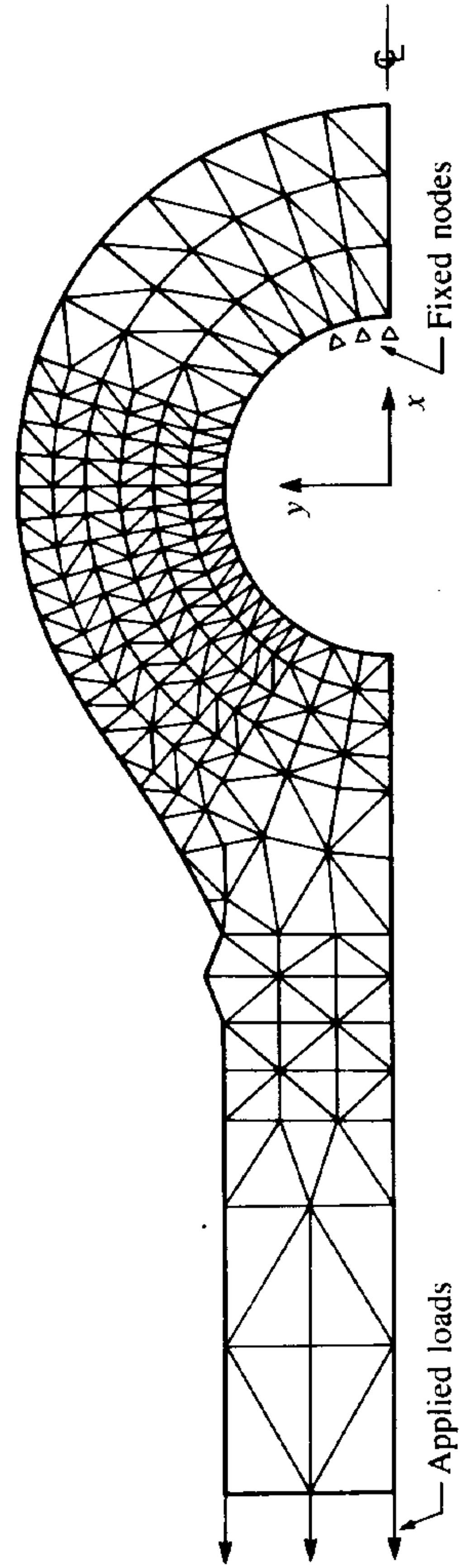
The next example shows a chimney stack section that is four form heights high (or a total of 32 ft high). The engineer used 584 beams to model the vertical and horizontal stiffeners making up the formwork, whereas 252 flat-plate elements were used to model the inner wooden form and the concrete shell.

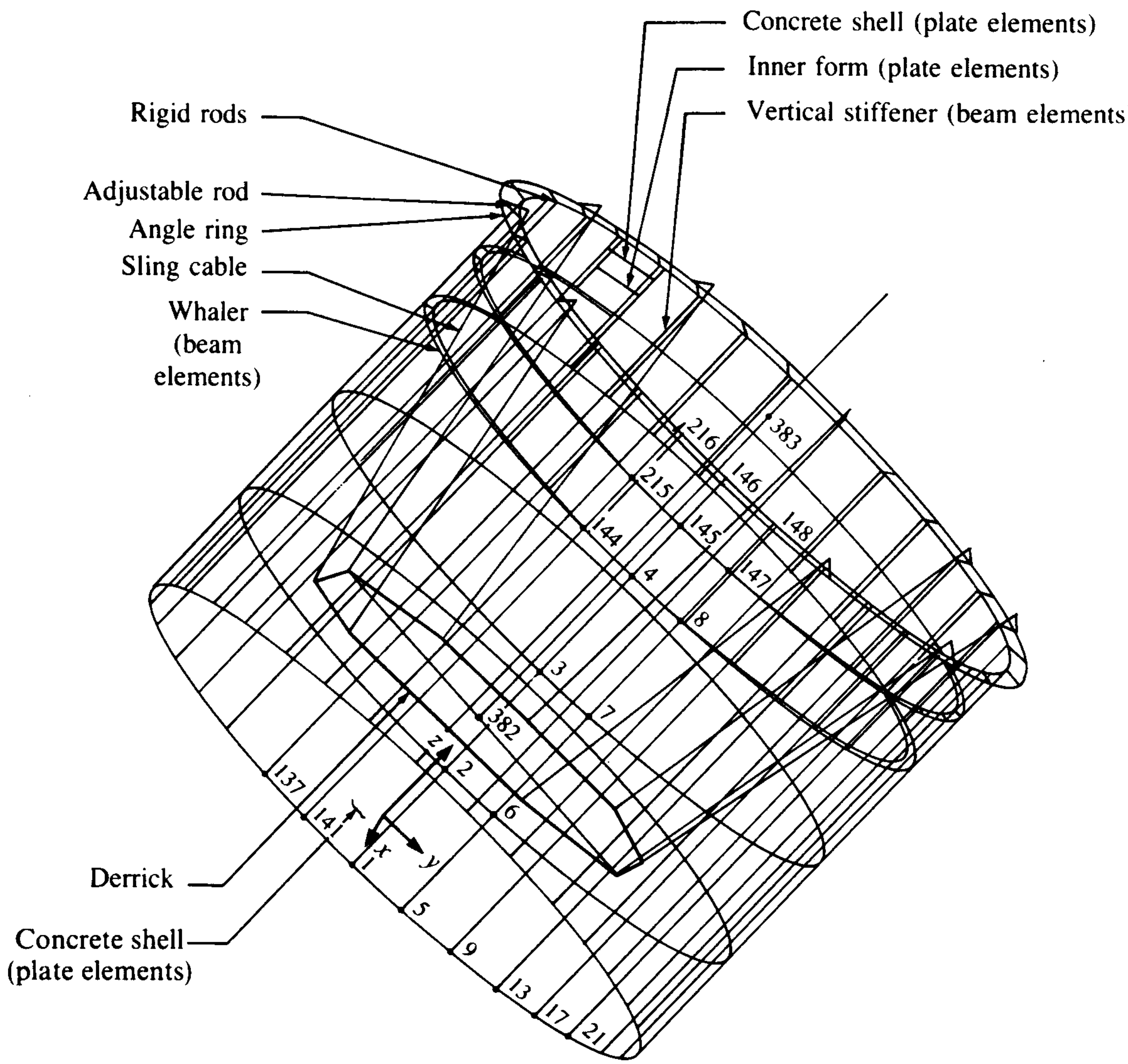
The next example shows the finite element discretized model of a proposed steel die used in a plastic film-making process. Two hundred forty axisymmetric elements were used to model the three-dimensional die.

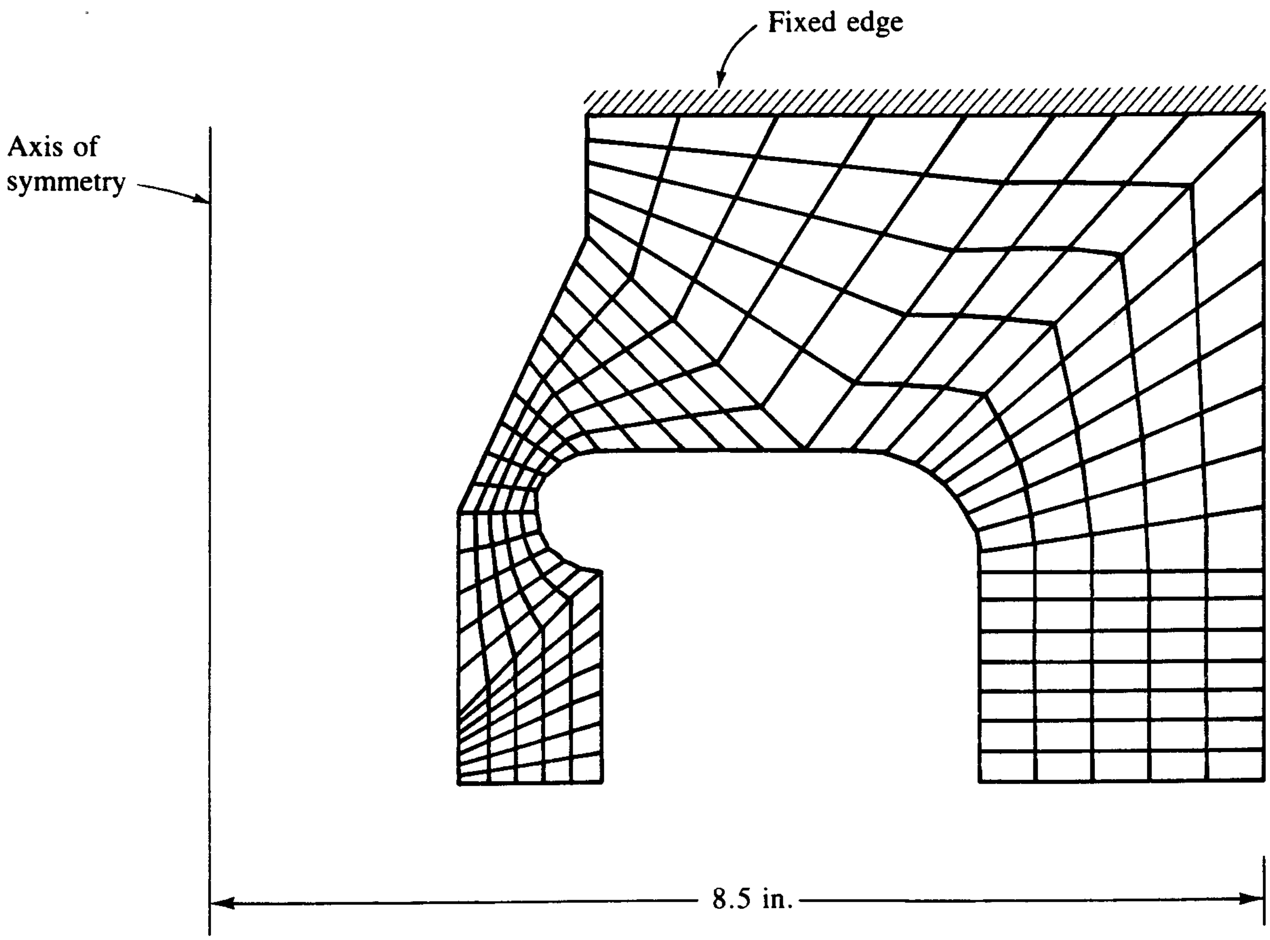

1
2
3
4
5
6
7
8
9
10
11
12
13
14
15
16
17
18
19
20
21
22
23
24
25
26
27
28
y
x
z
(1)
(2)
(3)
(4)
(5)
(6)
(7)
(8)
(9)
(10)
(11)
(12)
(13)
(14)
(15)
(16)
(17)
(18)
(19)
(20)
(21)
(22)
(23)
(24)
(25)
(26)
(27)
(28)
(29)
(30)
(31)
(32)
(33)
(34)
(35)
(36)
(37)
(38)
(39)
(40)
(41)
(42)
(43)
(44)
(45)
(46)
(47)
(48)



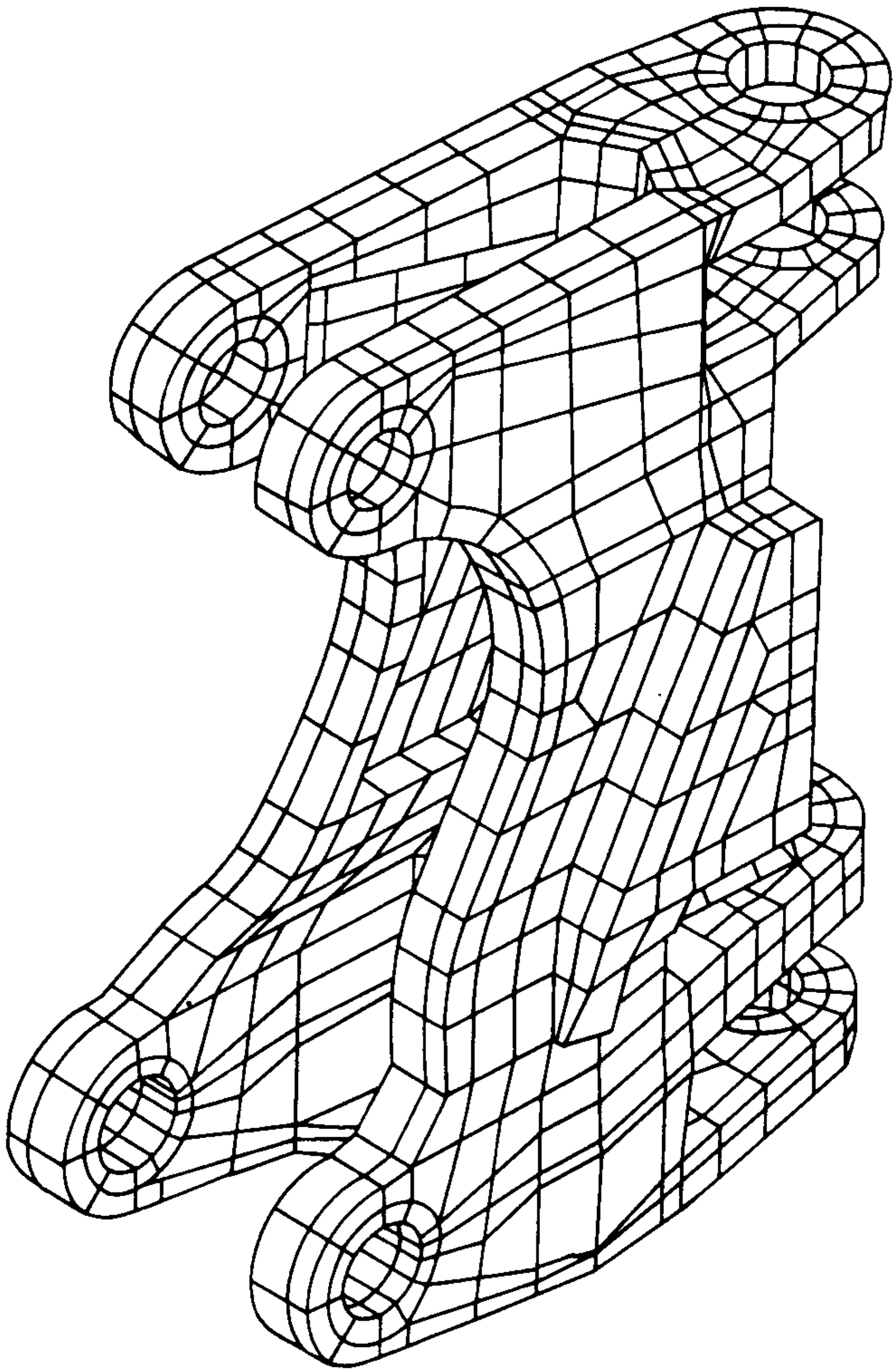
Fixed nodes along bottom


Fixed nodes
x
y
Applied loads


Concrete shell (plate elements)
Inner form (plate elements)
Vertical stiffener (beam elements)
Rigid rods
Adjustable rod
Angle ring
Sling cable
Whaler
(beam
elements)
Derrick
Concrete shell
(plate elements)
383
216
146
215
148
144
145
147
4
8
3
7
382
2
6
137
141
1
5
9
13
17
21
z
x
y


Fixed edge
Axis of
symmetry
8.5 in.

The next example illustrates the use of a three-dimensional solid element to model a swing casting for a backhoe frame. The three-dimensional hexahedral elements are necessary to model the irregularly shaped three-dimensional casting. Two-dimensional models certainly would not yield accurate engineering solutions to this problem.



Advantages of the Finite Element Method

The finite element method has been applied to numerous problems, both structural and non-structural. This method has a number of advantages that have made it very popular.

1. Model irregularly shaped bodies quite easily
2. Handle general load conditions without difficulty
3. Model bodies composed of several different materials because the element equations are evaluated individually
4. Handle unlimited numbers and kinds of boundary conditions
5. Vary the size of the elements to make it possible to use small elements where necessary
6. Alter the finite element model relatively easily and cheaply
7. Include dynamic effects
8. Handle nonlinear behavior existing with large deformations and nonlinear materials

The finite element method of structural analysis enables the designer to detect stress, vibration, and thermal problems during the design process and to evaluate design changes ***before*** the construction of a possible prototype.