FINITE ELEMENT METHOD

1. Basic Definition

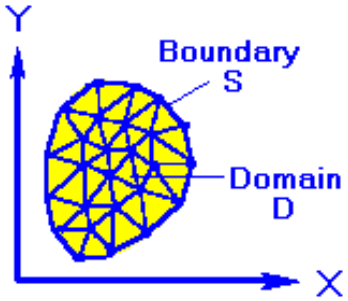
The finite element method is a numerical analysis technique used by engineers, scientists, and mathematicians to obtain solutions to the differential equations that describe, or approximately describe a wide variety of physical (and non-physical) problems. Physical problems range in diversity from solid, fluid and soil mechanics, to electromagnetism or dynamics.

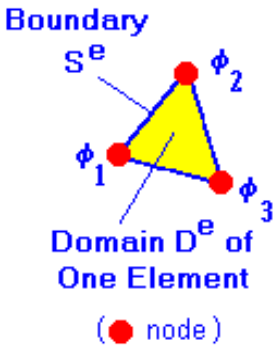
The underlying premise of the method states that a complicated domain can be sub-divided into a series of smaller regions in which the differential equations are approximately solved. By assembling the set of equations for each region, the behavior over the entire problem domain is determined.

Each region is referred to as an **element** and the process of subdividing a domain into a finite number of elements is referred to as **discretization.** Elements are connected at specific points, called **nodes**, and the assembly process requires that the solution be **continuous** along common boundaries of adjacent elements.

2. Discretization Using Finite Elements

Using the finite element method, the solution domain is discretized into smaller regions called **elements**, and the solution is determined in terms of discrete values of some primary field variables ϕ (e.g. displacements in x, y z directions) at the nodes. The number of unknown primary field variables at a node is the degree of freedom at that node. For example, the discretized domain comprised of triangular shaped elements is shown below left: In this example each node has one degree of freedom.





The governing differential equation is now applied to the domain of a single element (above right). At the element level, the solution to the governing equation is replaced by a continuous function approximating the distribution of ϕ over the element domain De, expressed in terms of the unknown nodal values ϕ_1, ϕ_2, ϕ_3 of the solution ϕ.

A system of equations in terms of ϕ_1, ϕ_2, and ϕ_3 can then be formulated for the element.

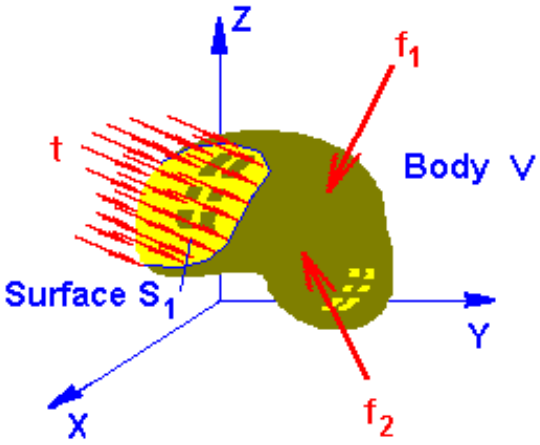
Once the element equations have been determined, the elements are assembled to form the entire domain D. The solution $\phi(x,y)$ to the problem becomes a **piecewise approximation**, expressed in terms of the nodal values of ϕ. A system of linear algebraic equations results from the assembly procedure. For practical engineering problems, it is not uncommon for the size of the system of equations to be in the thousands, making a digital computer a necessary tool for finding the solution.

3. Basic Theory of Finite Element Analysis (Structural Analysis)

Consider a general three-dimensional body, as shown below. External **point forces**, **f**, and distributed loads, **t**, called **tractions** act on a portion of the outer surface, **S_1**, of the body. This is

shown in the figure below:

Stresses within the solid body are a result of external forces that act on the surface of the body (called surface forces) or forces that act throughout the volume (called body forces). Examples of body forces include gravity and acceleration.



Under the influence of the applied forces, the body will deform. It is assumed that the body behaves elastically and returns to its initial configuration when the applied loads are taken away.

A measure of the relative deformation of the solid body is referred to as **strain**. In a Cartesian system, the components of strain are defined as shown in the figure below:

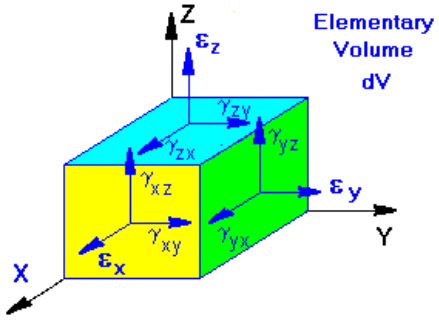
$$\varepsilon_x=\frac{\partial u}{\partial x}\ ,\ \varepsilon_y=\frac{\partial v}{\partial y}\ ,\ \varepsilon_z=\frac{\partial w}{\partial z}$$ **Normal Strains**

(1)

$$\gamma_{xy}=\gamma_{yx}=\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}$$

$$\gamma_{xz}=\gamma_{zx}=\frac{\partial u}{\partial z}+\frac{\partial w}{\partial x}$$ **Shear Strains**

$$\gamma_{yz}=\gamma_{zy}=\frac{\partial v}{\partial z}+\frac{\partial w}{\partial y}$$



At a given point, P, within the body, the strain can be calculated as functions of the u, v, w displacement components. Assuming that the strains are sufficiently small such that second order terms can be neglected, yields the strain displacement equations. The **normal strains** ε_x, ε_y, and ε_z are defined as the unit elongation of the body at a point in the direction of the respective x, y, z coordinate axes. The **shearing strains** measure the distortion of the angle between the various planes. For example, γ_{xy} measures the rotational distortion of the x-z and y-z planes. In general one can write the relationship in a matrix form:

$$\epsilon(x,y,z) = \mathbf{B}\,\delta(x,y,z)$$

The state of stress can be denoted by the **normal stresses** σ_x, σ_y, σ_z, and six components of **shear stress**. In a Cartesian coordinate system, these components are configured on an element of volume as shown in the figure below, and related to the strains by, e.g. Hooke's law.

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

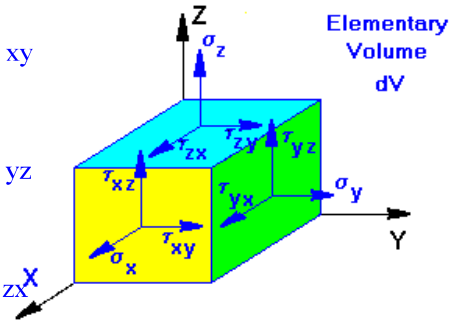
$$\varepsilon_y=\frac{\sigma_y}{E}-\nu\cdot\frac{\sigma_x}{E}-\nu\cdot\frac{\sigma_z}{E}$$

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

$$\gamma_{xy}=\frac{1}{G}\cdot\tau_{xy}$$

$$\gamma_{yz}=\frac{1}{G}\cdot\tau_{yz}$$

$$\gamma_{zx}=\frac{1}{G}\cdot\tau_{zx}$$



where E is the modulus of elasticity, ν is Poisson's ratio, and G is the modulus of rigidity. The more general stress-strain relationship that allows for different material properties can be written in a matrix form:

$$\sigma(x,y,z) = \mathbf{C}\,\epsilon(x,y,z)$$

Additional material properties, such as piezoelectric properties can be included by additional terms in the stress-strain relationship equations.

3.1 Minimum Potential Energy

For a given displacement function, strains can be calculated throughout the body from which stresses are derived using the constitutive laws. Clearly, a different displacement function will yield a different stress distribution.

An infinite number of geometrically possible displacement functions exist for the same body but there is one, unique displacement function that will physically describe the deformation due to a set of forces and satisfy the equilibrium of forces. This unique function can be determined by considering the principle of minimum potential energy.

Let Π represent the total potential energy of the system expressed as: $\Pi = U - V$

where U represents the **internal strain energy** and V represents the **external work** done on the body by the applied forces. The principle of minimum potential energy states that the displacement (u,v,w) that ensures equilibrium throughout the body will yield a minimum potential energy.

The total strain energy of the continuum, U, is defined as the volume integral:

$$U = \frac{1}{2} \cdot \int_{V} \int \int \boldsymbol{\varepsilon}^{\mathrm{I}} \cdot \boldsymbol{\sigma} \, dV$$

Using the general constitutive relationships between the stress and strain, and the strain and the displacement functions, one can express the total strain energy as a function of displacement:

$$U = \frac{1}{2} \cdot \int_{V} \int \int \underline{\boldsymbol{\delta}}(x,y,z)^{\mathrm{I}} \cdot \mathbf{B}^{\mathrm{I}} \cdot \mathbf{C} \cdot \mathbf{B} \cdot \underline{\boldsymbol{\delta}}\,(x,y,z)\, dV$$

The total work done by the external forces is equal to the sum of the products of each force and its respective displacement at the point of application. For the continuum under consideration:

$$V = \int_{S_1} \int \underline{\boldsymbol{\delta}}(x,y,z)^{\mathrm{I}} \cdot \mathbf{t}(x,y,z)\, dS_1 + \sum_{i=1}^{n_f} \left(\mathbf{d}_i^{\mathrm{I}} \cdot \mathbf{f}_i \right)$$

where

$\underline{\delta}(x,y,z)$ is the exact displacement function on the surface S1,
t(x,y,z) is the vector of surface tractions,
fi is the ith external force vector,
di is the displacement vector at the location where **f**i is applied, and
nf is the number of externally applied point forces.

Combining the equations give the total potential energy functional:

$$\pi = \frac{1}{2}\cdot\int_{V}\int\int \underline{\delta}(x,y,z)^{T}\cdot B^{T}\cdot C\cdot B\cdot\underline{\delta}\,(x,y,z)\,dV \quad + \ldots$$
$$- \int_{S_1}\int\int \underline{\delta}(x,y,z)^{T}\cdot t(x,y,z)\,dS_1 - \sum_{i:=1}^{n_f}\left(d_i^{T}\cdot f_i\right)$$

Now the principle of minimum potential energy can be stated as:

> Out of all geometrically possible displacement functions $\underline{\delta}(x,y,z)$ the one which minimizes the total potential energy, Π, is the displacement solution that will satisfy equilibrium, and will be the actual displacement due to the applied forces.

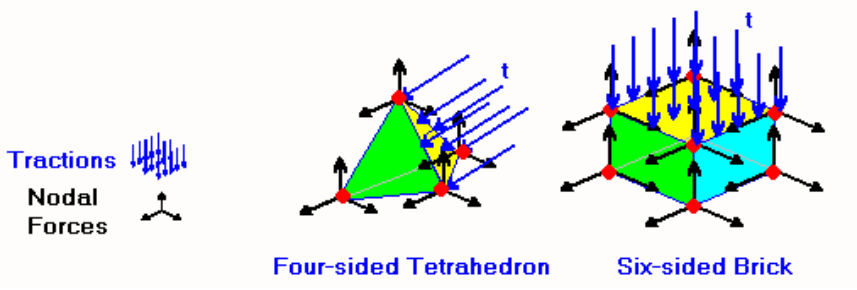
Thus, a displacement function that will minimize the functional, Π, is desired. Extremization of functionals falls within the field of **variational calculus**. In most cases an exact function is impossible to determine, necessitating the use of approximate numerical methods. The minimization of Π in a a finite element formulation is carried out using the energy approach. The finite element method develops the equations from simple element shapes, in which the unknowns of the solution become the displacements at the nodes.

3.2 Stiffness Matrix

If a continuum is divided into elements such that continuity of displacements between elements is enforced, then the total potential energy is equal to the sum of the individual energies of each element. For m number of elements, the total potential energy can be stated as:

$$\Pi = \sum_{e=1..m} \Pi^{(e)}$$

Examining the total potential energy for one element, the following figure presents two types of solid elements: tetrahedron and brick. The external forces acting on them are classified as either concentrated forces at the nodes or tractions on the outer surfaces of the element. For greater clarity, only tractions are shown acting on a single surface.



The exact displacement function, $\underline{\delta}$**e**(x,y,z), across the element is approximated as an interpolation

between the nodal displacements:

$$\underline{\delta}_{\underline{e}}(x,y,z) \approx N(x,y,z)\cdot\delta_e$$

where **N**(x,y,z) is the matrix of shape functions and δ**e** is the unknown vector of nodal displacements. Rewriting the total potential energy for an element using this equation:

$$\pi^{(e)} \approx \frac{1}{2}\cdot\iiint_{V^e} \delta_e{}^{I}\cdot N(x,y,z)^{I}\cdot B^{I}\cdot C\cdot B\cdot N(x,y,z)\cdot\delta_e \, dV^e - \iint_{S_1^e} \delta_e{}^{I}\cdot N(x,y,z)^{I}\cdot t(x,y,z)\, dS_1^e - \delta_e{}^{I}\cdot f_e$$

where Ve represents the volume of the element and S1e represents the surface on which tractions are applied. The last term of the equation is the dot product of the nodal displacement and force vectors, δ**e** and **fe**, respectively.

To minimize the total potential energy, Π, of the entire continuum, $\Pi^{(e)}$ must be minimized for each element. Seeking a set of nodal displacements for each element will minimize $\Pi^{(e)}$. Observe that the functional, $\Pi^{(e)}$ is a function only of the nodal displacements. Using calculus of variations, an extremization of $\Pi^{(e)}$ occurs when the vector of the first partial derivatives with respect to δ**e** is zero:

$$\frac{\partial \pi^{(e)}}{\partial \delta_e} = 0$$

Differentiating the energy function with respect to δe gives:

$$\frac{\partial \pi^{(e)}}{\partial \delta_e} = \iiint_{V^e} N(x,y,z)^{I}\cdot B^{I}\cdot C\cdot B\cdot N(x,y,z)\cdot\delta_e \, dV^e - \iint_{S_1^e} N(x,y,z)^{I}\cdot t(x,y,z)\, dS_1^e - f_e$$

Since δ**e** is not a function of position, it may be taken outside of the integral. Setting the above equation equal to zero with some manipulation yields the element equation corresponding to the minimum energy

$$K_e\cdot\delta_e = f_e + \iint_{S_1^e} N(x,y,z)^{I}\cdot t(x,y,z)\, dS_1^e$$

where **Ke** is the element stiffness matrix:

$$K_e = \int_{V^e}\int\int N(x,y,z)^T \cdot B^T \cdot C \cdot B \cdot N(x,y,z)\, dV^e$$

Since strain energy cannot be negative, the stiffness matrix must be **positive definite**. In addition, **Ke** is symmetric due to the **symmetric** nature of the matrix product in the equation.

The above equations are applicable to all elastic finite elements. Upon assembly of the element equations, the traction term (involving **t**(x,y,z)) will only appear for elements whose surfaces are exterior to the assembled continuum.

This section has derived the static finite element equations in a general form, applicable to any type of element in an elastic continuum. Using the principle of minimum total potential energy, a functional, $\Pi_{(e)}$, was developed in terms of the nodal displacements.

For a specific element shape function, there exists a set of nodal displacements which will minimize the functional at both the element level and the overall continuum level. The calculus of variations enabled the energy equation to be reduced to a set of simultaneous equations with the nodal displacements as the unknown quantities.

3.3 Dynamic Analysis - Mass and Viscous Damping Matrices

Variational methods enabled the derivation of finite element equations for all physical problems that could be expressed in terms of a functional such as energy. One area of particular interest is the application of finite elements in dynamics and vibrations. If the frequency of the excitation applied to a structure is higher than about one-third of the structure's lowest natural frequency of vibration or if the structure vibrates freely, then inertia becomes important and one need to include the mass effect and the damping effect into the finite element formulation. Equations that govern the dynamic response of a structure or medium can be derived by similar variational principle, i.e. by requiring the work of external forces to be absorbed by the work of internal, inertial, and viscous forces for any kinematically admissible motion. The inertial work done by a virtual displacement is given by

$$\int_{V_e}\int\int d\delta_e^T \, \rho \, \frac{\partial^2 \delta_e}{\partial t^2} \, d\, V$$

and that by the viscous force is

$$\int_{V_e}\int\int d\delta_e^T \, \zeta \, \frac{\partial \delta_e}{\partial t} \, d\, V$$

Following similar procedure as for the stiffness matrix, the mass and viscous damping matrices can be found:

$$M_e \frac{\partial^2 \delta_e}{\partial t^2} + C_e \frac{\partial \delta_e}{\partial t} + K_e \delta_e = f_e + \int_{S_1}\int N(x,y,z)^T \, t(x,y,z) \, dS_1^e$$

where

$$\boldsymbol{M_e} = \iiint_{V_e} \rho \; \boldsymbol{N}(x,y,z)^T \; \boldsymbol{N}(x,y,z) \; dV$$

and

$$\boldsymbol{C_e} = \iiint_{V_e} \zeta \; \boldsymbol{N}(x,y,z)^T \; \boldsymbol{N}(x,y,z) \; dV$$

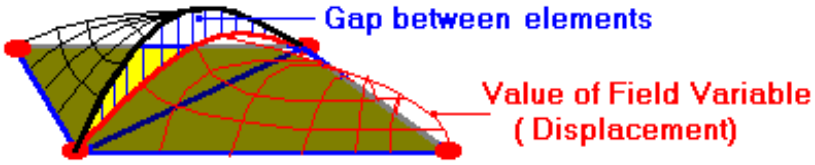
when the same shape function is used to generate the stiffness, mass, and viscous matrices, the element is called isoparametric.

4. Basic Steps of a Finite Element Analysis

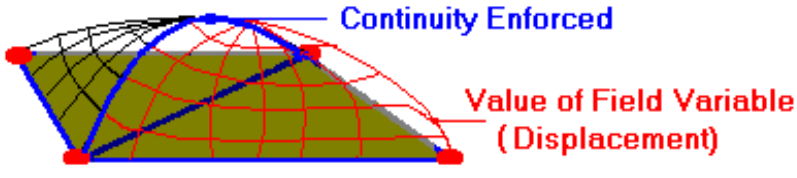
Step 1: Discretize the Continuum

For a continuous domain there is no natural subdivision, so the mesh pattern will appear somewhat arbitrary. The continuum would be replaced by a series of simple, interconnected elements whose force-displacement characteristics are relatively easy to compute.

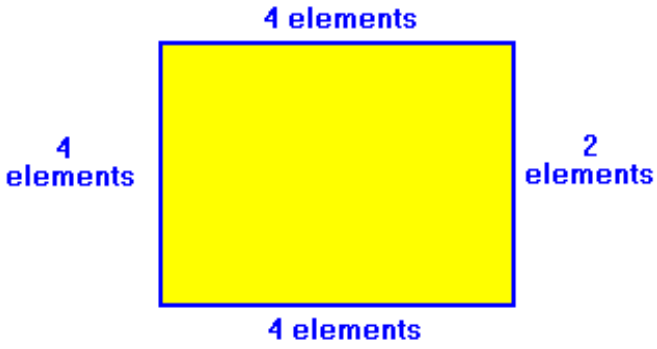
In reality, these elements are connected to each other along their boundaries but in order to perform a theoretical approximation, the assumption is made that the elements are connected **only** at their nodes. The figure at the right shows the deformation of two elements with nodal compatibility. Notice the excessive flexibility of the mesh.



This type of flexibility significantly reduces the accuracy of the approximation. However, the **continuity requirement** which forces the variation of the field variable along an element interface to be the same for adjacent elements is required within the finite element method. Finite elements of a continuum are special types of elements that are constrained to maintain overall continuity of the assemblage.

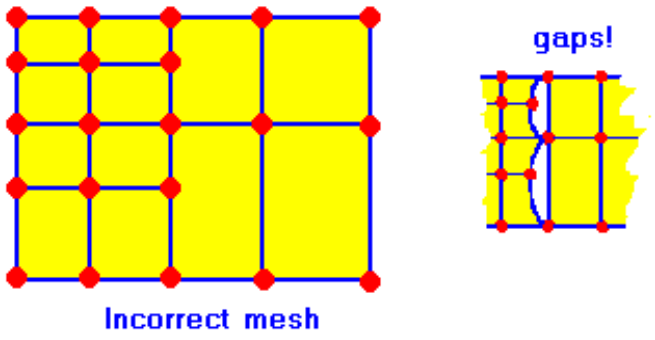


A wide variety of elements types in one, two, and three dimensions are well-established and documented. It is up to the analyst to determine not only which types of elements are appropriate for the problem at hand, but also the density required to sufficiently approximate the solution. Engineering judgment is essential.

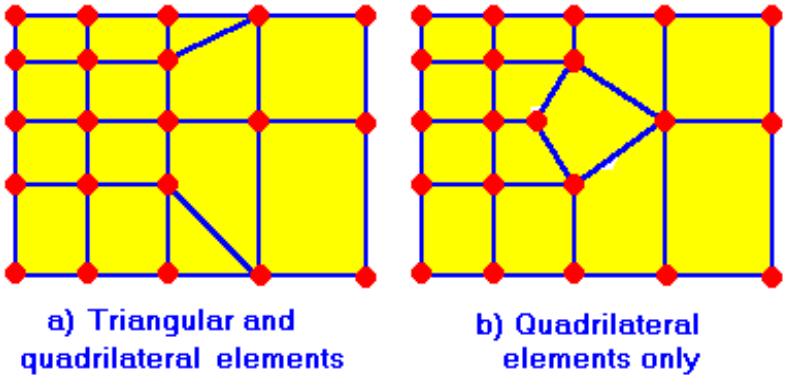


A finite element mesh of a continuous region must not have any gaps due to improperly connected elements. For example, suppose one is interested in discretizing a rectangular region having the following number of elements on the boundaries:

Within the domain, a transition from four elements on the left to only two elements on the right is required. An incorrect mesh using four-noded quadrilateral elements is shown below:



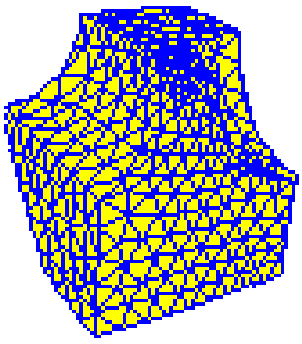
The gaps in the above mesh result from attempting to connect a node to an edge of an element, rather than to another element's node. In order to eliminate the gaps, elements must be connected **node to node**. Two correct possibilities are shown below:



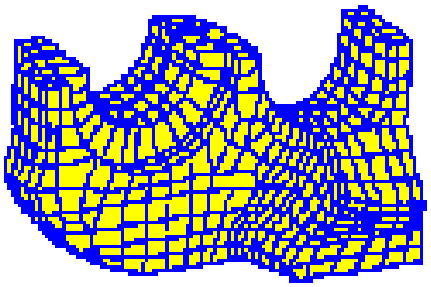
Correct Meshes

Discretization of irregular shaped regions has traditionally been performed manually, from sketches. Today, state-of-the-art software packages automate the meshing process. Caution is still advised, however, with any mesh-generation package, and the user's judgment and experience still remain important.

Once a finite element mesh has been created, it must be checked to ensure that each element satisfies certain criteria for acceptability, for example distortion, which may produce spurious results.



Automatic Mesh Generation



Manual Mesh Generation

Step 2: Select Interpolation Functions

Once an element shape has been chosen, the analyst must determine how the variation of the field variable across the element domain is to be represented or approximated. In most cases, a

polynomial interpolation function is used. The number of nodes assigned to an element dictates the order of the interpolation function which can be used.

Interpolation functions are also referred to as **shape functions** or **approximating functions.**

If the analyst is using an existing finite element package, then most likely the choice of interpolation function is implicit in the choice of element type.

Shape Functions (Element Types)

The finite element method describes the behavior of a continuum using a discretization of the continuum into smaller, manageable regions called **elements**. The unknown field variable (or variables, e.g. displacements) for which the solution is sought is expressed in terms of a discrete number of unknowns at each **node**. Since a continuous solution is being represented, the solution must also be continuous throughout each element.

The method of approximating the solution across each element is referred to as **element interpolation**. The degree to which the approximate solution is sufficient to accurately model the problem is affected by the type of interpolation, or the **shape function**, used.

The simplest method of approximation is to assume a linear distribution of the unknown function within the element domain. Most one-dimensional functions can be represented as a series of straight lines. The smaller the line segments, the more accurate the solution will be. Conversely, the larger the line segments, the less accurate the solution will be. Computational costs, on the other hand, may increase significantly as the number of elements in a model increases. Thus, it may be more practical to use fewer elements, and ones that have a higher order of interpolation. Another consideration is that in most analysis, the solution may require continuity of the first derivatives (e.g. strain) of the field variable as well as the variable (e.g. displacement) itself.

In the description of the basic theory, the finite element method was used to find a solution to a displacement vector field using an interpolation between the discrete displacements at each node. In particular, the variation of displacement across a 2-dimensional (x,y, only) single element can be expressed as

$$\underline{u}_e(x,y) \approx u_e(x,y) = \sum_i N(x,y)_i \cdot u_i$$

where the underlined function, $\underline{ue}(x,y)$, denotes the exact x component of displacement and, ui, denotes the x displacement of node i. The displacements can be generalized to include any type of scalar field quantity, ϕ.

Approximation to the exact function is defined as $\phi(x,y)$:

$$\underline{\phi}(x,y) \approx \phi(x,y)$$

Thus, for a given element, the approximation can be defined as the following summation:

$$\phi_e(x,y) = \sum_i N(x,y)_i \cdot \phi_{e_i}$$

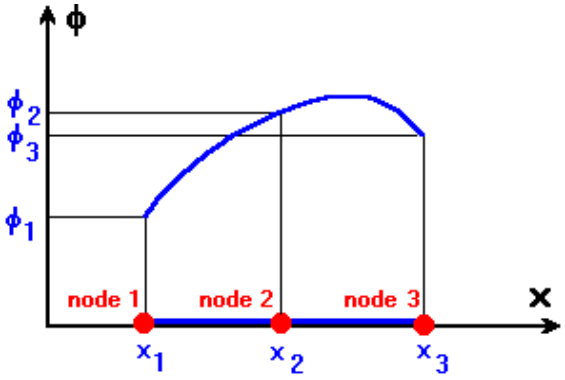
i = 1 .. number of nodes per element

where ϕe(x,y) is the field approximation across the element, N(x,y)i, is the ith shape function and ϕ**e**i is the unknown field quantity at node i. (Unless otherwise specified, functions will be represented in x and y.)

Simple Example: Quadratic Interpolation in One Dimension

Let us define a quadratic polynomial in one dimension:

$$\phi(x) = \alpha_1 x^2 + \alpha_2 x + \alpha_3$$



α1, α2, and α3 are the coefficients that determine the shape of the function ϕ. Since there are three coefficients in the equation, three nodes are required for a line element to fit the polynomial. This type of element is referred to as a quadratic line element.

It is possible to simply solve a set of equations for α1, α2, and α3 by evaluating the equation at x1, x2, and x3. Alternatively, it is possible to derive shape functions for each node by re-expressing the equation as

$$\phi(x) = N_1(x) \cdot \phi_1 + N_2(x) \cdot \phi_2 + N_3(x) \cdot \phi_3$$

where the three quadratic shape functions N1, N2, and N3 are:

$$N_1(x) := a_1 x^2 + b_1 x + c_1$$

$$N_2(x) := a_2 x^2 + b_2 x + c_2$$

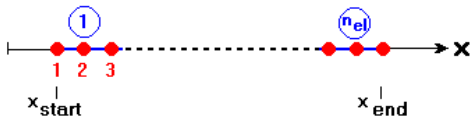
$$N_3(x) := a_3 x^2 + b_3 x + c_3$$

$$\begin{bmatrix} N_1(x) \\ N_2(x) \\ N_3(x) \end{bmatrix} = \begin{bmatrix} a_1 & b_1 & c_1 \\ a_2 & b_2 & c_2 \\ a_3 & b_3 & c_3 \end{bmatrix} \cdot \begin{bmatrix} x^2 \\ x \\ 1 \end{bmatrix}$$

To find the coefficients, we use the fact that the shape function for a particular node will have the value 1 at that node and zero at the other two nodes, thus yielding the following set of equations:

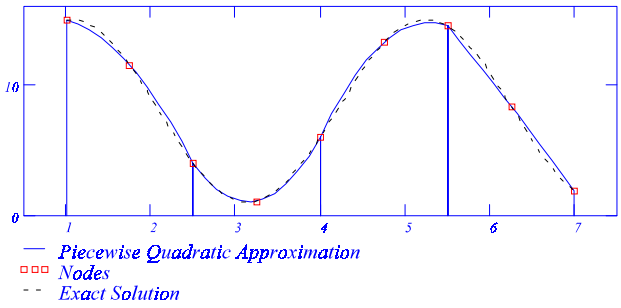
$$\begin{pmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{pmatrix} = \begin{bmatrix} x_1^2 & x_2^2 & x_3^2 \\ x_1 & x_2 & x_3 \\ 1 & 1 & 1 \end{bmatrix} \cdot \begin{bmatrix} a_1 & b_1 & c_1 \\ a_2 & b_2 & c_2 \\ a_3 & b_3 & c_3 \end{bmatrix}$$

From which the coefficients can be solved.



The above quadratic shape functions were derived over the domain of a three-noded element. These will then have to be connected to adjacent elements to form the complete model.

As an example, the above quadratic interpolation will be applied to a series of connected, three-noded elements as shown in the figure. An example solution of the field variable over the entire domain is shown below:



The figure clearly demonstrates a piecewise quadratic interpolation between the values of the field variable at each node. It is **piecewise** because the distribution is calculated one piece at a time, element by element, and the value of $\phi(x)$ within each element is fitted to a quadratic polynomial.

The accuracy of the approximation can be improved by changing the number of finite elements in the domain, in the expense of computation time, of course. Rather than increasing the mesh density, one can use a higher-order shape function to increase the accuracy of the approximation. The procedure can be generalized to higher orders and to more than one dimension.

Derivatives of Shape Functions

It has been shown that the variation of the field variable across a line element is represented by the matrix equation

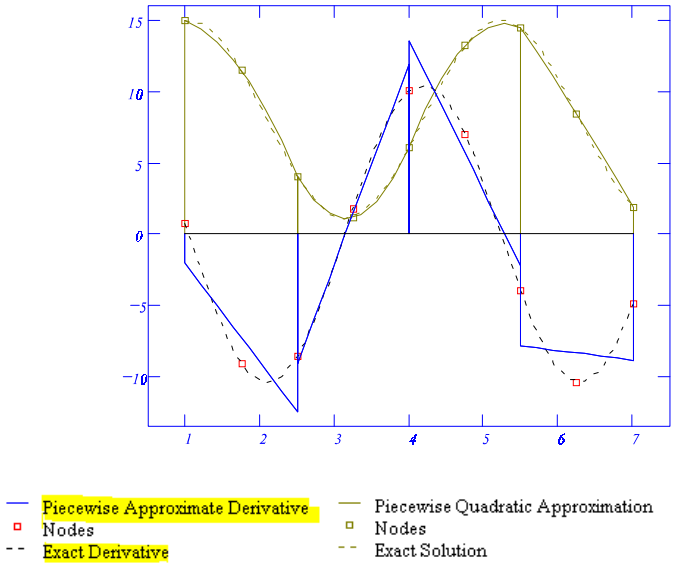
$$\phi(x) = N(x)^T \cdot \phi_e$$

where $\mathbf{N}(x)$ is the vector of shape functions and $\phi_\mathbf{e}$ represents the field variable at each of the nodes. Differentiating with respect to the spatial variable x yields

$$\frac{d}{dx}\phi(x) = \left(\frac{d}{dx}N(x)\right)^T \cdot \phi_e$$

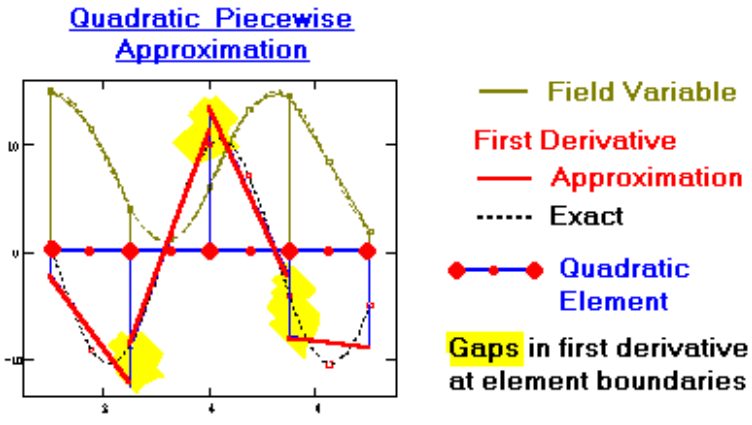
since $\phi_\mathbf{e}$ is constant at each of the nodes.

In the above section, the continuity of the field variable at element boundaries was discussed in the context of displacements. From a physical point of view, continuous displacements were necessary to prevent holes or tears in the finite element mesh. In applications such as plate bending, the slope or derivative of the displacement must also be continuous at element interfaces. Continuity requirements are further discussed in this section by examining the derivative of the piecewise approximation for the quadratic case used in the previous example. The above graph shows the variation of the first derivative of the field variable over the entire domain as shown in previous graph. In a quadratic approximation, the first derivatives are approximated using piecewise linear segments. Similar to the linear piecewise approximation, discontinuities of first derivatives exist at each element interface.



Continuity Requirements

In an earlier section, it was stated that the displacement along an element boundary had to be the same for all elements sharing that boundary. If this were not true, then gaps would introduce excessive and unrealistic flexibility into the structure. This imposed constraint on the displacement is referred to as the continuity requirement.



Now, consider the quadratic piecewise approximation for four elements shown.

The approximation to the field variable is continuous but the first derivative of the approximation is discontinuous at the element boundaries. The discontinuities are highlighted as gaps in the figure. If elements of this type are used to model the bending of beams, then the gaps in the above figure would represent discontinuities in the shape of the deformed beam.

Such an approximation is said to have C^0 continuity. If the first derivatives were continuous between elements, then the approximation would have C^1 continuity. In general, a function is C^n continuous if the derivatives up to the nth order are continuous.

Continuity is important from a physical perspective (for example to prevent gaps in displacements or slopes) and, also, from a mathematical point of view. It can be shown that the finite element solution will converge to the exact solution as the number of elements increases provided that two

conditions are satisfied:

Compatibility: 1) C^{n-1} continuity exists at the element interfaces , and

Completeness: 2) there is C^n continuity of the field variable within the element,

where n is the highest-order derivative that appears in the element equations.

These two conditions are usually referred to as the **compatibility** and **completeness requirement**, respectively.

Note that it is simple to construct polynomial shape functions with C^0 continuity while higher-order continuity requirements are more difficult to construct. Generally specific shape functions, or finite element types, are designed for specific applications to improve accuracy, e.g. beam and plate elements.

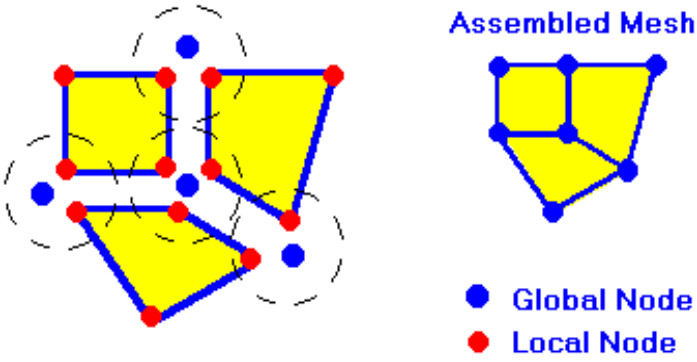
Step 3: Find the Element Properties

Once the interpolation functions have been chosen, the field variable in the domain of the element is approximated in terms of discrete values at the nodes. Consequently, a system of equations is formed which expresses the element properties in terms of quantities at the nodes.

For example, in a structural analysis, the element equations relate the nodal forces to the nodal displacements.

Step 4: Assemble the Element Properties

The assembly procedure combines each element approximation of the field variable (as defined in the previous steps) to form a piecewise approximation of the behavior over the entire solution domain. Assembly is accomplished using the following basic rule of compatibility:



The value of the field variable at a node must be the same for each element that shares that node.

This step is handled automatically by the finite element package.

Step 5: Apply the Boundary Conditions

The global system of equations created in the previous step cannot be solved, pending application of the boundary conditions. Mathematically, before applying the boundary conditions, the system of equations is indeterminate and does not have a unique solution. In the same way that a structure must be physically fixed to the ground to prevent it from moving when a force is applied, a node must also be fixed to the ground.

In finite element terminology, a node is **prescribed** if the value of the field variable is already known. For example, a prescribed nodal displacement of zero is assigned to fixed supports of a structure. In a thermal analysis, a temperature is prescribed at one or more nodes. Usually these nodes lie on the boundaries of the finite element mesh.

Step 6: Solve the System of Equations

Once the boundary conditions have been applied to the assembled matrix of equations, standard numerical techniques can be used to solve for the unknown field variable at each node. If the system of equations is linear, a Gaussian elimination or Cholesky decomposition algorithm can be used.

In structural analyses, the matrix equations generated by the finite element process are often sparsely populated and symmetrical. Many solution methods make use of these properties to provide fast and efficient computation algorithms which are now implemented in nearly all finite element packages.

In dynamic analysis, the matrix equation can be reduced to a sub-set by taking advantage of the fact that the lower natural frequencies are governed mainly by the structural parts that have the highest mass to stiffness ratio.

If the set of equations is non-linear, then the solution procedure is more difficult to obtain.

The choice of solution method is very much dependent on the size of the problem as well as the type of analysis.

Step 7: Make Additional Computations

Based upon the solution to the field variable, additional calculations include the computation of principal stress, strain energy and dynamic responses in a structural analysis.

In dynamic analysis with viscous only damping, it is convenient to calculate the modal frequencies and mode shapes of the structure first and calculate the dynamic response later using modal analysis - the modal result is used to diagonalise the mass, damping and stiffness matrices to form a set of de-copuled, single degree-of-freedom modal equations of motion, which can be solved easily by analytical means. However this method does not work if the damping is significantly not viscous, or in fluid-structure coupled analysis.

Summary

Among the seven steps, it is arguable that the first two steps: the discretisation of the structure and the choice of element types (shape functions) largely determine the validity and hence the accuracy of the finite element solution.

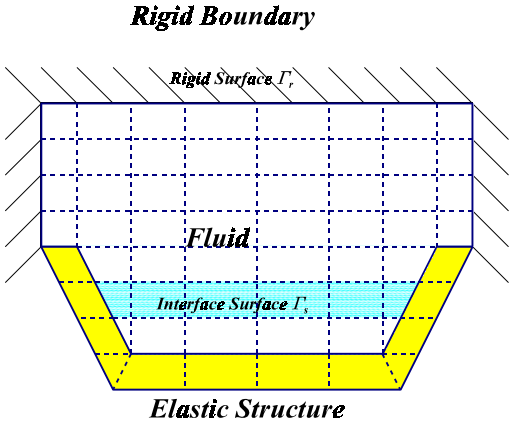
Steps 3 to 7 are usually more straightforward.

Finite element method is a very flexible and powerful method for solving differential equations involving field variables over a continuous but finite domain, which represents most engineering applications. However the details of the numerical method is highly complex. The situation is further complicated by the large variety of software implementations of the method each of which has its own interpretation of the method and specific formulation of features. It will usually take a significant amount of time for a user to get use to and to understand a particular finite element software package before usable results can be generated.

5. Acoustic Finite Elements

Acoustic finite elements are most commonly used for modelling finite domain acoustic problems. For infinite regions (e.g. free field) an artificial truncation boundary with an approximate boundary impedance or some special semi-infinite elements can be used, although in this case the boundary element method tends to be more efficient.

Formulation of the method can be based on discretising the pressure field. or the displacement field. The process also uses the variational procedure based on the virtual work done, but the stress-strain relationship in an elastic medium is replaced by the acoustic pressure-displacement relationship and the element properties are defined by the fluid's density and bulk modulus. Absorbent materials can also be modelled.



Using a pressure based formulation, the degree of freedom at each acoustic finite element node corresponds to the acoustic pressure field variable, while the "excitation" corresponds to the particle displacement. The finite element matrix equation over the fluid domain can be derived:

$$[M_a]\left\{\frac{\partial^2 p}{\partial t^2}\right\} + [K_a]\{p\} - \iint_{\Gamma_s} [N]^T \left\{\frac{\partial^2 u_n}{\partial t^2}\right\} d\Gamma = 0$$

where M_a is the pressure based acoustic "mass" matrix

$$[M_a] = \frac{1}{\kappa} \iiint_V [N]^T [N]\, dV$$

and K_a is the pressure based acoustic "stiffness" matrix

$$[K_a] = \frac{1}{\rho} \iiint_V [N]^T [N]\, dV$$

Since the normal displacement $u_n = \underline{n}^T [N_s] \{u\}$, we can write

$$[M_a]\left\{\frac{\partial^2 p}{\partial t^2}\right\} + [K_a]\{p\} - [T]\left\{\frac{\partial^2 u}{\partial t^2}\right\} = 0$$

where

$$[T] = \iint_{\Gamma_s} [N]^T \, \underline{n}^T \, [N_s] \, d\Gamma$$

Fluid-structure Coupling

Over the rigid boundary the displacement is zero. On the interface boundary between the fluid and an elastic structure, the displacement in the surface normal direction must be continuous, and the acoustic pressure will appears as surface force on the structure.

Denoting {**F**} as the total external forces except the acoustic pressure, acting on the elastic structure, then the finite element matrix equation over the domain of the elastic structure is

$$[M] \left\{\frac{\partial^2 u}{\partial t^2}\right\} + [K] \{u\} + [T]^T \{p\} = F$$

This equation can then be coupled to the equation over the fluid medium to give the fluid-structure coupled finite element equation

$$\begin{bmatrix} [K] & [T]^T \\ [0] & [K_a] \end{bmatrix} \cdot \begin{Bmatrix} \{u\} \\ \{p\} \end{Bmatrix} + \begin{bmatrix} [M] & [0] \\ -[T] & [M_a] \end{bmatrix} \cdot \begin{Bmatrix} \left\{\frac{\partial^2 u}{\partial t^2}\right\} \\ \left\{\frac{\partial^2 p}{\partial t^2}\right\} \end{Bmatrix} = \begin{Bmatrix} \{F\} \\ \{0\} \end{Bmatrix}$$

which can then be solved numerically.

One consequence of coupling the fluid to the structure is that the coefficient matrices are no longer symmetrical or positively defined. As a result the efficient matrix solution algorithms used in pure structural analysis cannot be used here. Alternatively the matrices can be symmetrised. Generally a coupled problem is a lot more time consuming to solve than a pure structural problem of the same size.

REFERENCES

Cook R.D, Malkus D.S, and Plesha M. E., *Concepts and Applications of Finite Element Analysis*, 3rd Ed., John Wiley and Sons, 1989.

Huebner K.H., Thornton E.A., *The Finite Element Method for Engineers*, Second Edition, John Wiley and Sons, 1982.

Petyt M., "Finite Element Techniques for acoustics", *NOISE AND VIBRATION*, ed. R.G. White and J.G. Wlaker, Ellis Horwood Ltd., 1982.